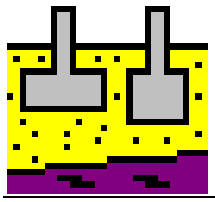
Analysis of foundation settlements to DIN 4019

GGU-SETTLE

VERSION 7



Last revision: November 2023

Technical implementation, layout and sales: Civilserve GmbH, Steinfeld

Contents:

List of Figures:

List of Tables:

1 Preface

The **GGU-SETTLE** program allows analysis of foundation settlements including mutual influence using rectangular or triangular foundations. The stress course at the foundation base is linearly variable in all directions. Settlement depressions, lines of equal settlement, stress distribution etc. can be calculated and presented. The influence of vibrodisplacement compaction after Priebe can be investigated.

The use of triangular foundations is always of advantage if the load geometry cannot be described in a satisfactory manner using rectangular foundations. This is advisable for settlement analyses of, e.g., the following systems:

- circular foundations
- annular foundations
- loading (e.g. for a landfill) which is extremely difficult to model with rectangular foundations
- cone-shaped loading figures.

The theoretical principles are taken from the "Grundbautaschenbuch" (Foundation Engineering Pocketbook), 1990, Volume 1. Further information on the subject of triangular foundations can be found in the following article which is available (currently in German, to be translated at a later date) through our distributor:

- Dr. Johann Buß,
  Setzungen und Spannungen unter "Dreiecksfundamenten"
  (Settlements and Stresses below Triangular Foundations),
  Geotechnik 22 (1999) No. 1

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 6.1).

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 system has been used in a large number of projects by renowned consultancies and institutes, and has been thoroughly tested. No faults have been found. Nevertheless, liability for completeness and correctness of the program and the manual, and for any damage resulting from incompleteness or incorrectness, cannot be accepted.

2 Capabilities

The **GGU-SETTLE** program has the following capabilities:

- Analysis of foundation settlements according to DIN 4019.
- Generation of triangular foundations which are combined in a mosaic-like manner in order to model any possible type of loading.
- Consideration of mutual influence of foundations.
- Analysis of settlements at any point inside or outside of the foundations.
- Analysis of settlements at a given depth.
- Analysis of settlements at a given layer base.
- Analysis of settlements of individual layers.
- Analysis of subgrade reaction modulus courses.
- Complete input of system geometry, if desired, using the mouse.
- 50 soil layers.
- 1000 rectangular foundations or 15,000 triangular foundations.
- Excavation unloading can be considered.
- For a given excavation unloading, the constrained modulus for reloading can be considered for the resulting reloading.
- Limiting depth calculation via x % of the overburden stress or via a multiple of the foundation width or as fixed value.
- Generation of footing foundations, circular foundations and annular foundations.
- Calculation and presentation of settlement depressions.
- Calculation and presentation of stress distributions (also as contour plan = stress bulb).
- Calculation and presentation of lines of equal settlements in plan and in any desired vertical section.
- Presentation of calculation results in tables.
- Presentation of a legend with soil properties.
- Presentation of a legend with general information on the basis for calculations.
- **Mini-CAD** system for free labelling and drawing of graphical elements in the output sheet.
- Free definition of page size.

3 Licence protection

To protect the GGU software from unauthorised access, each GGU program is equipped with the ***CodeMeter software protection system*** from WIBU-Systems. Each GGU 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.

4 Language selection

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

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

5 Starting the program

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

- File
- Info

After clicking the "**File**" menu, either an existing system can be loaded by means of the "**Load**" menu item, or a new system can be entered using "**New**". After clicking "**File/New**" you will first see a dialog box for selecting rectangular or triangular foundations for the following analysis. Then, the menu bar will show ten menus:

- File
- Soils
- Mesh
- Foundations or Triangular foundations
- System
- Evaluation
- Special
- Graphics preferences
- Page size + margins
- Info

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

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

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

6 Tips and tricks

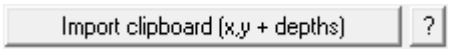
6.1 "?" and "Info" buttons

Reading of the manual can mostly be dispensed with, because

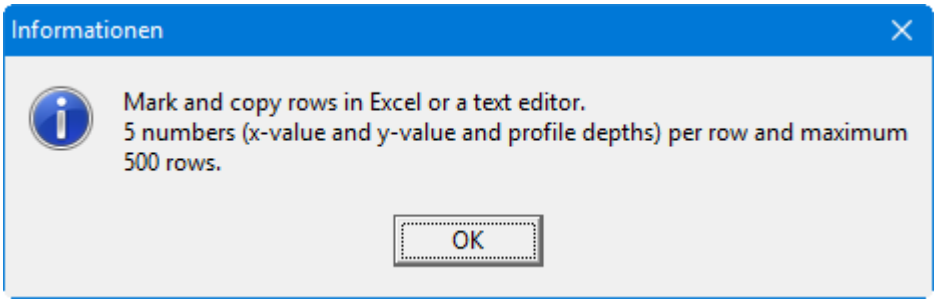
"**?**" buttons  and "**Info**" buttons 

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 "**Mesh/Edit nodes (via table)**" dialog box:

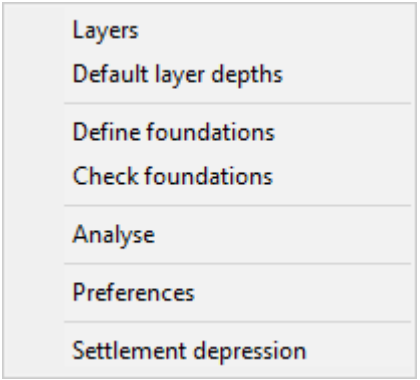


If you click on the question mark, you will see the following message box:



6.2 Keyboard and mouse

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



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

Any calculation and presentation can be prematurely cancelled using the right mouse button.

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

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

In input boxes for numerical values, you can also use common arithmetic operations to make adjustments (see Section 6.4).

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

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

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

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

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

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

6.3 Function keys

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

- [**Esc**] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [**F1**] opens the manual file.
- [**F2**] refreshes the screen without altering the current magnification.
- [**F3**] opens the menu item "**Soils/Edit layer depths**".
- [**F4**] opens the menu item "**System/Limiting depth**".
- [**F5**] opens the menu item "**System/Analyse**".
- [**F6**] opens the menu item "**System/Analysis options**".
- [**F7**] opens the menu item "**Soils/Determine layer depths**".
- [**F8**] opens the menu item "**(Triangular)Foundations/Check foundations**".
- [**F9**] activates the menu item "**Page size + margins/Auto-resize**".
- [**F11**] activates the menu item "**Graphics preferences/Move objects**".

6.4 Calculation functions in input boxes with numbers

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

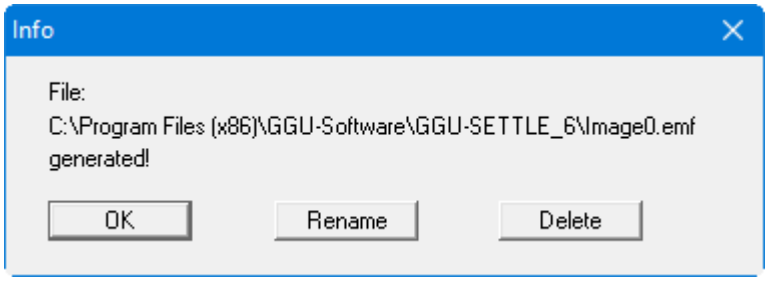
6.5 "Copy/print area" icon

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

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

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

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



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

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

7 Short introduction using worked examples

7.1 Program concept

In order to calculate foundation settlements, input of the constrained modulus profile is necessary, as well as the foundation data (width, length, foundation stress). Many comparable programs define this constrained modulus ***per foundation***. The **GGU-SETTLE** program takes a different, much more flexible path. The constrained modulus profiles are defined in a triangle mesh. At the nodes of this triangle mesh you can edit the thickness of the layers involved. After having defined the triangle mesh you can place the foundations within this mesh. During the following calculations the program determines the valid constrained modulus profile from linear interpolation within the triangle concerned. Even after calculations are complete, or after placing the foundations, the triangle mesh can be edited or supplemented.

In order to secure a sensible interpolation within the constrained modulus profile mesh, it is necessary for all nodes to have the same number of layers, and for all layers to have the same constrained modulus within the mesh. The thickness of the layers, however, can be varied. If your system has areas in which certain soil layers are not present (e.g. peat lenses only in certain areas), then simply assign these layers a thickness of '**0**' at the appropriate nodes.

As, from personal experience, the reading of user manuals is a chore, there will now follow a short description of the main program functions using the following worked examples. After studying these sections you will be in a position to carry out calculations with the **GGU-SETTLE** program. You can take the details of the program from the following chapters.

7.2 Example 1: Analysis of rectangular foundations

7.2.1 System data (Ex. 1)

For example, you know the constrained modulus profile of 5 boreholes (BP 1 to BP 5) from field investigations. The boreholes have the following coordinates:

Borehole	x [m]	y [m]
BP 1	0.0	21.0
BP 2	1.0	6.0
BP 3	12.5	15.0
BP 4	25.0	22.0
BP 5	24.0	4.0

Table 1 Borehole coordinates

We have a 3-layer system. The individual layers have the following soil properties:

Layer no.	Soil type	Constrained mod. initial loading [MN/m²]	Constrained mod. reloading [MN/m²]	Poisson's-ratio [-]	Unit weight of soil [kN/m³]
1	Silt	12	30	0.0	18
2	Sand	45	110	0.0	10
3	Clay, silty	6	15	0.0	11

Table 2 Soil types and soil properties

Unit weight input is only decisive if the limiting depth is to be calculated with a percentage value of the overburden stress. Constrained modulus input for reloading is only decisive when foundations with excavation unloading are being processed.

The layers at the individual boreholes, measured from ground level (GL), reach the following depths:

Borehole	Layer base 1 [m b. GL]	Layer base 2 [m b. GL]	Layer base 3 [m b. GL]
BP 1	2.00	3.50	15.00
BP 2	2.00	4.00	15.00
BP 3	2.00	5.00	15.00
BP 4	2.00	5.00	15.00
BP 5	2.00	4.00	15.00

Table 3 Layer depths

In the example, two foundations with the designations '**F1**' and '**F2**' are present.

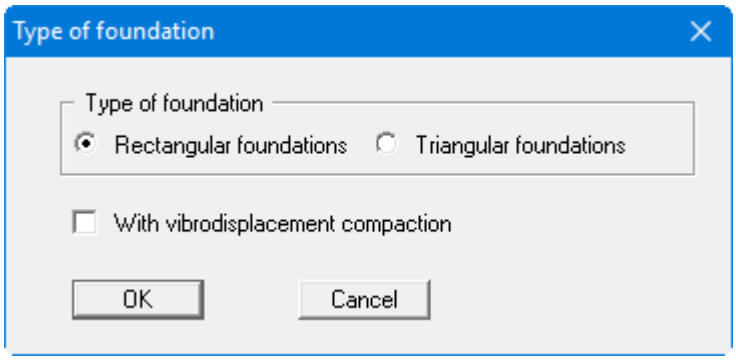
Foundation	x (left [m]	y (bottom) [m]	Foundation pressure [kN/m²]	Length [m]	Width [m]
F1	4.0	14.0	200.0	7.0	4.0
F2	14.0	8.0	200.0	3.0	3.0

Table 4 Foundation dimensions and pressure

Both foundations have their base at 0.8 m below ground level.

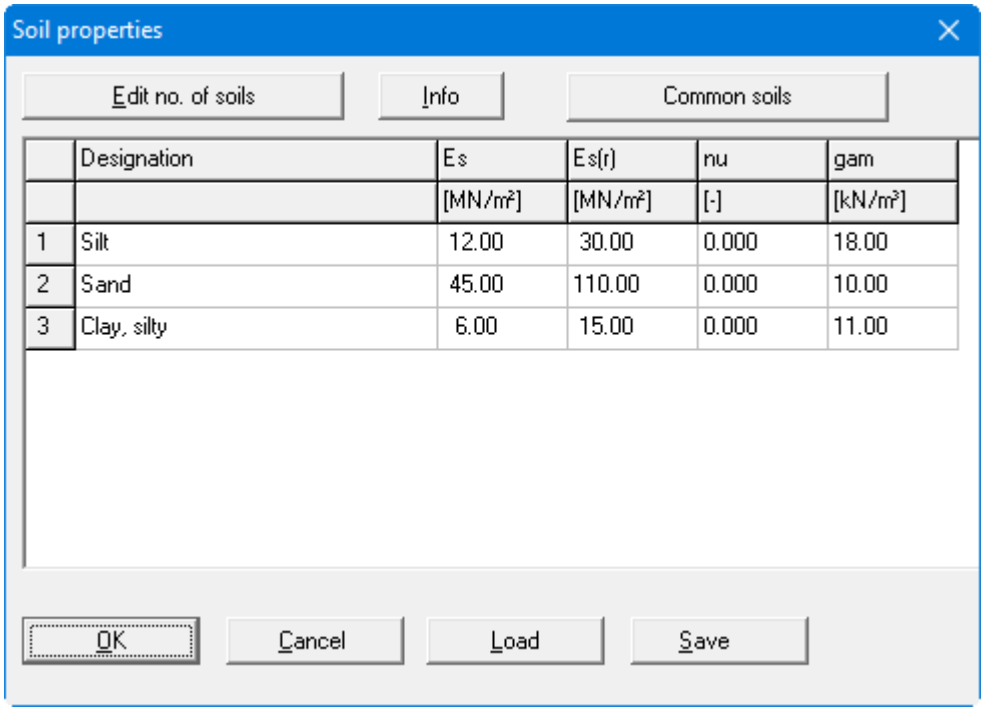
7.2.2 Define soils (Ex. 1)

First select the menu item "**File/New**" and activate the "**Rectangular foundations**" check box.



The first step in a calculation using the **GGU-SETTLE** program is the definition of a triangle mesh. The nodes of the triangle mesh are described by the borehole points. The borehole points are then connected to a triangle mesh, from which the program can interpolate the constrained modulus profile at any point. To define the borehole points proceed as follows:

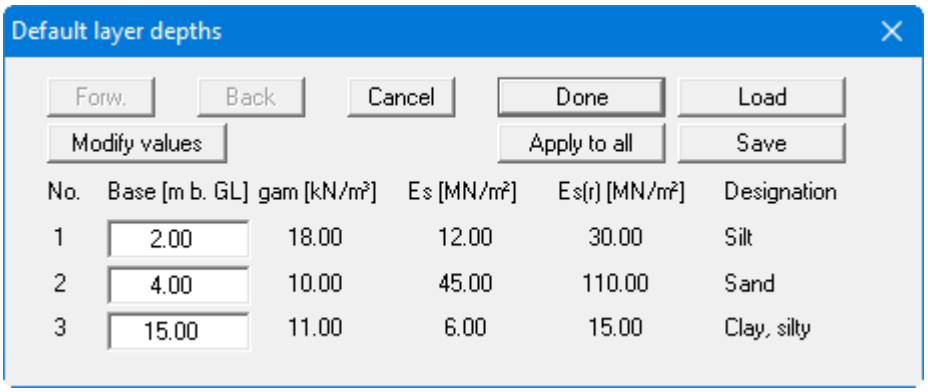
First, go to the menu item "**Soils/Layers**" and click the "**Edit no. of soils**" button. Enter '**3**' for the new number of layers. Three rows are then displayed for the layers in the following dialog box. Enter the soil properties from Table 2 (soil types and soil properties) as shown.



	Designation	Es	Es(r)	nu	gam
		[MN/m²]	[MN/m²]	[-]	[kN/m³]
1	Silt	12.00	30.00	0.000	18.00
2	Sand	45.00	110.00	0.000	10.00
3	Clay, silty	6.00	15.00	0.000	11.00

7.2.3 Define default layer depths (Ex. 1)

You should then select the menu item "**Soils/Default layer depths**" (not absolutely necessary).

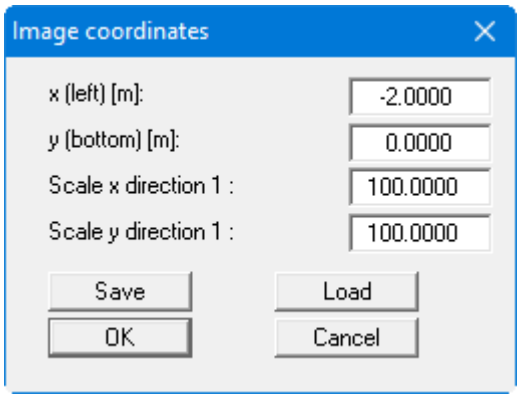


As you have defined three layers using the menu item "**Soils/Layers**", three layers will be shown in the dialog box. The corresponding soil properties are also shown for information purposes. The given depths can be edited to suit your wishes. Depths are entered as m below ground level. For reasons of clarity, the program assumes ground level to be the same for all nodes. During the following definition of triangular nodes, the layer depths are assigned to the appropriate nodes. After this, the layer depths for each node can be edited as wished. If the constrained modulus profile is equal for almost all nodes, you can save input work via the initial definition of default layer depths. The layer depths for the example were therefore entered in the dialog box based on Table 3.

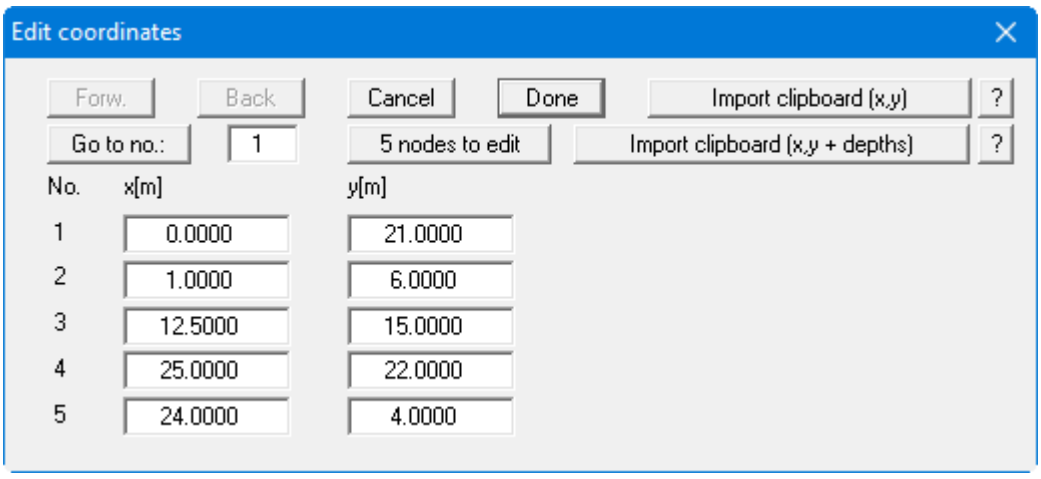
Using the "**Load**" button you can load a previously saved depth distribution. Using "**Save**" you can save the current depth distribution to a file, in order to have them available later, without renewed input. Using the "**Modify**" button, all existing depths can be increased or reduced by applying a constant. If you select the "**Apply to all**" button, all current triangle nodes will be assigned these depths. Using the "**Cancel**" button, you may leave the dialog box without accepting the alterations. You can also leave the dialog box with "**Done**". Any changes will be accepted.

7.2.4 Define triangle nodes

After this initial input you can determine the position of triangle nodes (borehole points). A coordinate system is visible on the screen. If the area displayed does not correspond to the plan area of your system, go to the menu item "**Page size + margins/Manual resize (editor)**" and enter the values for your system into the dialog box.



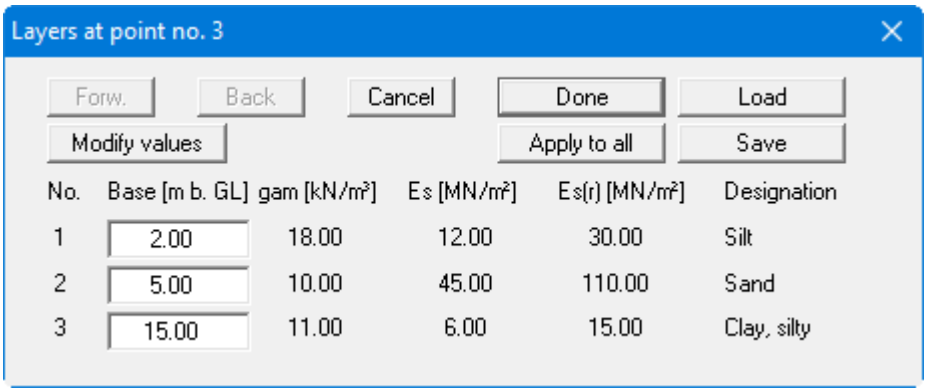
Then select the menu item "**Mesh/Define nodes**". Click, with the left mouse button, on the positions of the triangle nodes (= positions of points for which you know the constrained modulus profile). In the program status bar the current coordinates of the mouse pointer are shown. Erroneous input can be undone by clicking on the node with the right mouse button. If the presentation appears too small or too large, go to the menu item "**Evaluation/Preferences**" and enter a factor for the constrained modulus profile width and/or depth to suit your wishes. Alternatively to coordinate input using the mouse, you can enter the values in tabular form. Then select the menu item "**Mesh/Edit nodes (via table)**" and change the number of points to '**5**' using the "**x nodes to edit**" button.



The data for the example system based on Table 1 (coordinates of drilling points) are entered in the above dialog box. After entering the data go to the menu item "**Page size + margins/Auto-resize**" or alternatively [**F9**] in order to achieve a screen-filling visualisation of the constrained modulus profiles. If you already have the coordinates and/or layer depths available in a text editor or in Excel, they can be imported from the Windows clipboard using the two corresponding buttons. Information on this is provided via the two "**?**" buttons. By pressing the lower button, "**Import clipboard (x,y + depths)**", the program anticipates 2 columns for x and y and then as many columns as you have defined soil strata.

7.2.5 Adjust layer depths of nodes (Ex. 1)

After entering nodes, you can edit the depths at the nodes for the individual layers. For this, select the menu item "**Soils/Edit layer depths**". A double-click in the area of the node will then suffice to call up the following dialog box.



This box is absolutely identical to the box for default depths. The only difference is that any changes made are with reference to the current node. Depths are entered as m below ground level. Edit the input for the drilling points at which the layer depths deviate from the default section in accordance with Table 3 (BP 1, BP 3 and BP 4).

7.2.6 Define triangle mesh (Ex. 1)

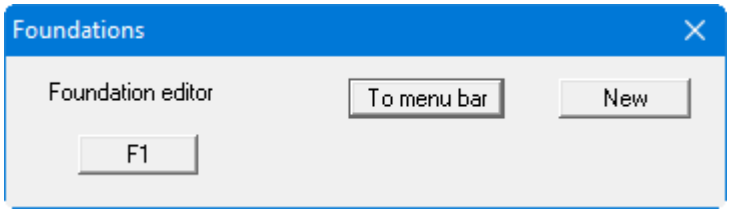
After you have defined at least 3 nodes (borehole points), you must connect these nodes to a triangle mesh to allow the program to interpolate when calculating. There are two possibilities:

- "**Mesh/Manual mesh**" menu item
  You must click on the three nodes to be connected to a triangle. Mistakenly created triangles can be deleted by clicking on the three nodes once again.
- "**Mesh/Automatic**" menu item
  The program carries out a so-called triangulation and connects all nodes to a triangle mesh. Subsequently, you still have the possibility of deleting triangles by using the menu item "**Mesh/Manual mesh**" and clicking on the three nodes of the triangle. If a triangle mesh is already present you will be asked, before triangulation begins, if the current mesh is to be deleted. Only in exceptional cases should you select "**Supplement**", as triangulation follows certain laws which may not allow sensible complementing of a current partial mesh.

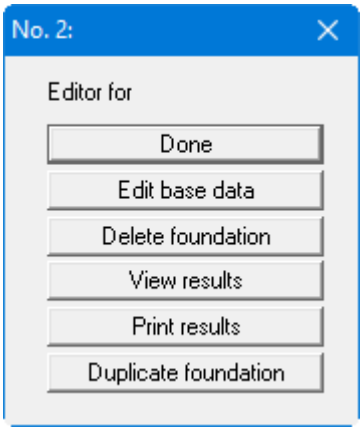
For the example select the "**Mesh/Automatic**" menu item.

7.2.7 Define rectangular foundations (Ex. 1)

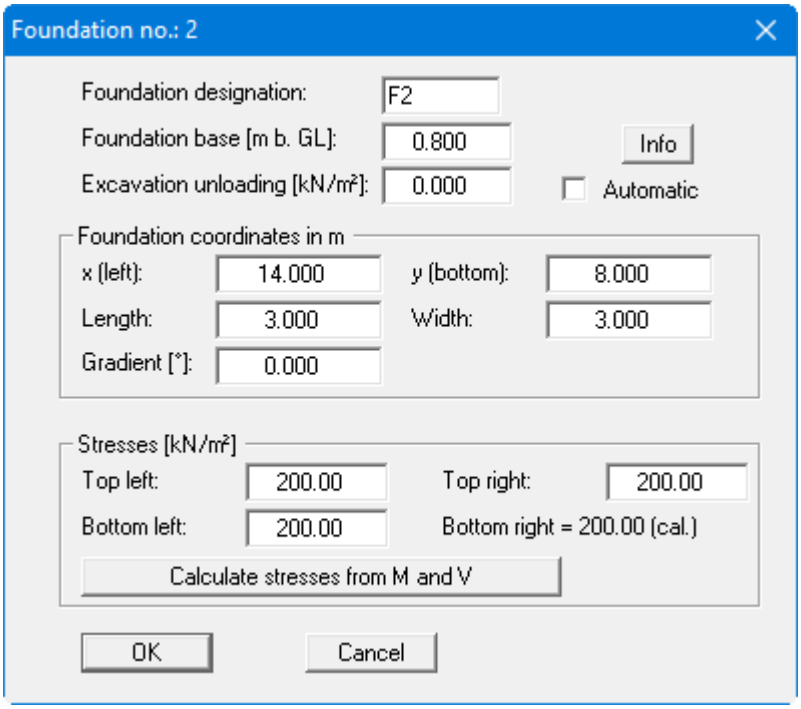
After defining the constrained modulus profile distribution throughout the triangle mesh, you can enter the foundations. For this, select the menu item "**Foundations/Define**".



In the dialog box shown the first foundation ("**F1**" button) is already entered. Click the "**New**" button. You will then see the following dialog box:



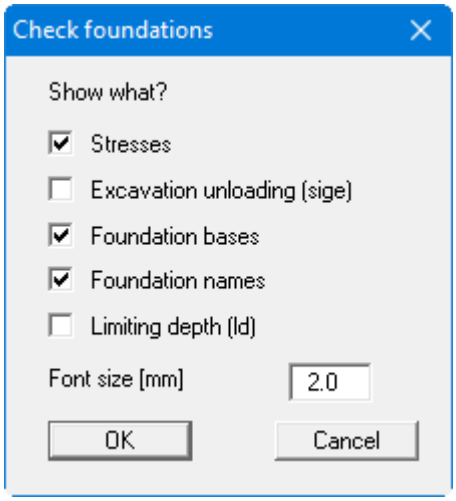
After clicking the "**Edit base data**" button the data for the second foundation can be edited in accordance with Table 4. Descriptions of the remaining buttons can be found in Section 9.4.1.



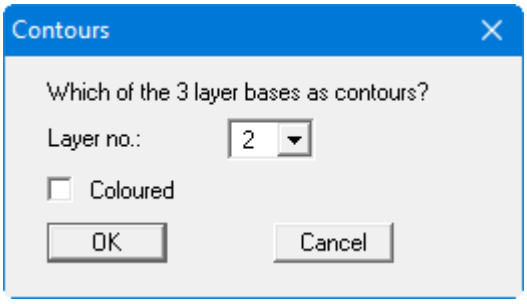
After the foundation designation you enter the depth of the foundation base (FB), the excavation unloading, the foundation dimensions and the corner stresses. For linear stress distribution, input of three stress values is sufficient. The fourth value then results automatically. If foundations are already defined, you can also reach this dialog box by double-clicking in the foundation surround on the screen.

7.2.8 Check input data (Ex. 1)

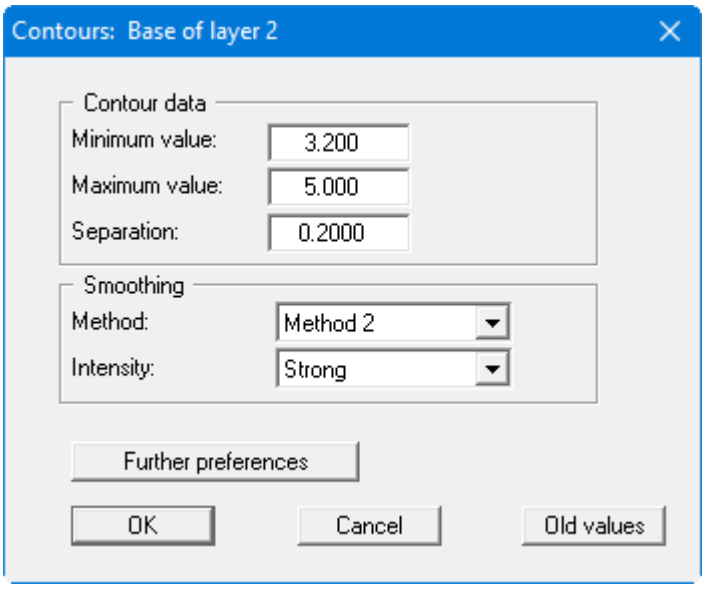
The input for the foundations can be checked using the menu item "**Foundations/Check foundations**". Alternatively, you can press function key [F8]. The foundations are visualised with corner stresses. In the dialog box, specify the additional input data to be visualised:



A contour diagram of the layer base can be generated to check the layer depths that have been entered. To do this, select the menu item "**Soils/Layer base contours**".



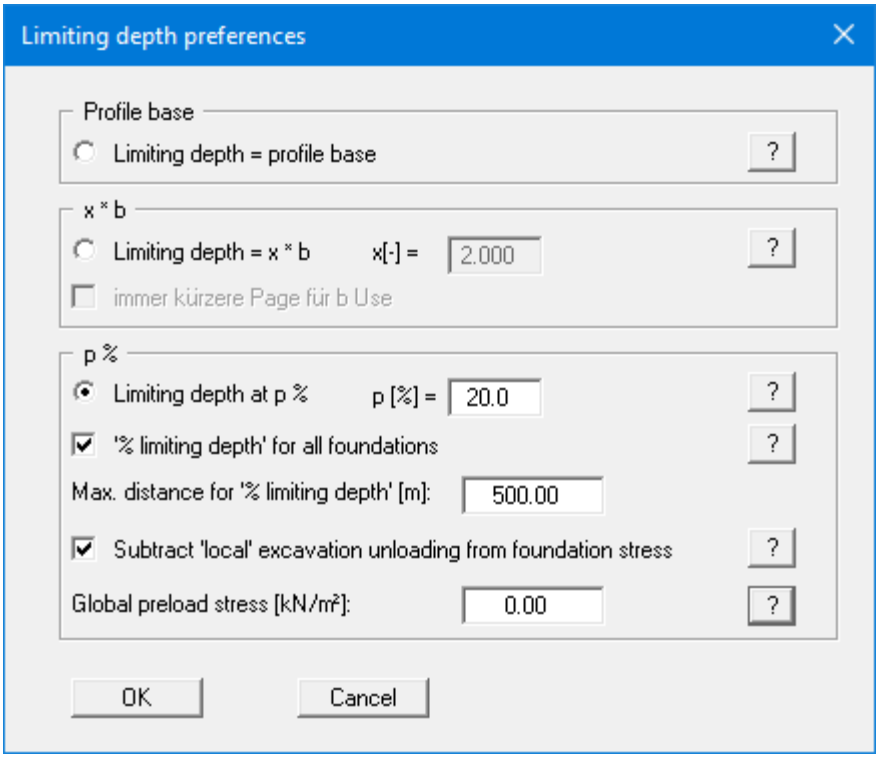
Enter the number of the required layer. Deactivate the "**Coloured**" check box if a normal line diagram is required for contour visualisation. Confirm with "**OK**".



In the upper part of the dialog box you can influence the number of contour lines with the input after "**Separation**". If desired, change to a different smoothing out procedure. After exiting the box using "**OK**" the lines of equal layer base elevation are visualised on the screen.

7.2.9 Specify limiting depth (Ex. 1)

Calculating rectangular foundations the program can determine the limiting depth using three different procedures. Select the menu item "**System/Limiting depth**" or the function key [**F4**].

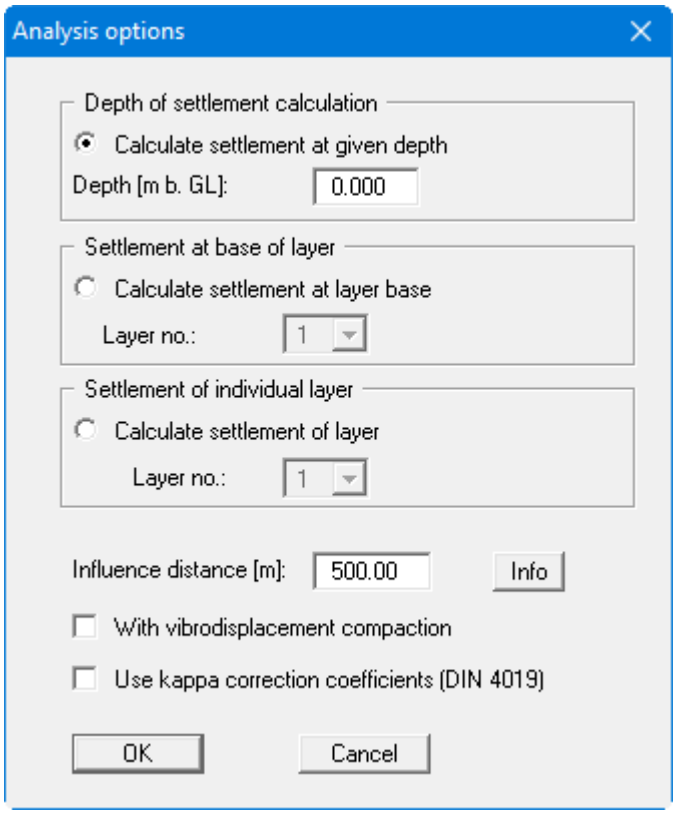


- "**Limiting depth = profile base**"
  The settlements are calculated to the base of the defined soil profile.
- "**Limiting depth = x * b**"
  The settlements are calculated to a depth of x * foundation width, but not deeper than the profile base.
- "**Limiting depth at p %**"
  The settlements are calculated to a depth at which the overburden stress * p from the soil weight corresponds to the foundation stress, but not deeper than the profile base. If foundations are close to one another (e.g. with circular foundations), you should, additionally, activate the "**% limiting depth for all foundations**" check box, as otherwise too shallow limiting depths will be calculated. When determining the limiting depth with p %, any excavation unloading for the foundation concerned can be subtracted from the average foundation stress, which usually makes sense. More information on excavation unloading and global preload stress can be had by clicking on the "**Info**" button in the above dialog box. The limiting depth will be calculated in the centre of the rectangular foundation.

If the settlement is to be calculated outside of the foundation concerned, the limiting depth of the foundation which creates the settlement stress is also valid. Stresses above the foundation base generate tensional stresses and will therefore not be considered.

7.2.10 Specify type of settlement analysis (Ex. 1)

With the menu item "**System/Analysis options**" you can specify the type of settlement to be calculated. Alternatively, you can access this menu item using function key [F6]. In principle, there are three different types:



- "**Calculate settlement at given depth**"
  This is usually the standard case, together with a depth of 0.0 m (= settlement at the ground level). As the area of the foundation is indeformable, the settlement at the foundation base, including for foundations with the base below the ground level, will be calculated. If a settlement depth > 0.0 is entered the settlement at this depth will be calculated.
- "**Calculate settlement at layer base**"
  As the layers need not necessarily be horizontally arranged, you have the possibility with this calculation method of following the influence of settlements at certain layer boundaries. To do this, activate this check box and enter the layer number.
- "**Calculate settlement of a layer**"
  Quite often the deformation component of an individual layer is of interest. To do this, activate this check box and enter the layer number.
- In the lower part of the dialog box, you can define a so-called influence distance. The influence of a foundation on the current settlement value will only be calculated if the distance to the foundation centre is smaller than the influence distance. If you have defined a lot of foundations the calculation time can become quite long, as, for each point at which the settlement is calculated, the influence of all foundations is considered. The influence of a foundation on the current settlement decreases with increasing distance. You can thus achieve a reduction of calculation durationusing the influence distance. A reliable statement on influence distances cannot be given, as the foundation loading also plays a large role. If you are unsure, use, as in the above dialog box, a very large value and if you are calculating a lot of foundations allow yourself a coffee during calculations.

Any calculation and presentation can be prematurely cancelled using the right mouse button.

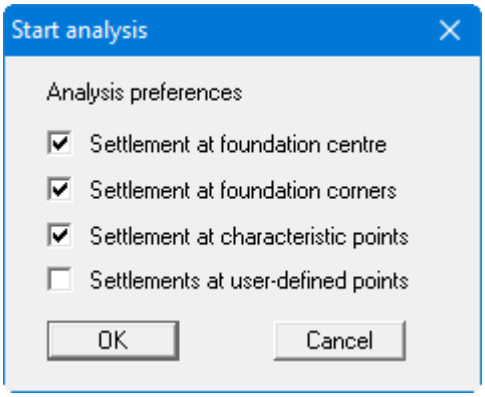
- "**With vibrodisplacement compaction**"
  Activate vibrodisplacement compaction adoption after Priebe (Heinz J. Priebe, *Die Bemessung von Rüttelstopfverdichtung*, Ground Engineering, December 1995) using this check box.
- "**Use kappa correction coefficients (DIN 4019)**"
  By activating this check box the kappa correction coefficients compliant with DIN 4019 Table 1 can be adopted for settlement analysis.

7.2.11 Calculate foundation settlements (Ex. 1)

After defining a foundation you can have the system calculated. In principle, we can differentiate between analysis of foundation settlements and the analysis of user-defined settlements. The two calculations can also be mixed. With foundation settlement we mean the settlement at special points on the foundation:

- settlement in the foundation centre
- settlement at the four characteristic points
- settlement at the four foundation corners.

If you are only interested in settlement at the above described foundations points, select the "**System/Analyse**" menu item or press the function key [**F5**].



Activate the check box(es) of choice and click "**OK**". You may see a note on the specified special preferences first. Acknowledge the notes and start the analysis. Information on analysis progress is displayed in the status bar. Once analysis is complete the foundation settlements are entered in the foundation centre, in the corners and in the characteristic points, depending on the preferences specified above.

If you have calculated a system and then edited any input values (e.g. foundation dimensions, foundation stress, type of limiting depth calculation, etc.), the program deletes all calculated settlements, as they are then no longer valid for the altered system. If you have already defined user-defined settlement points within the framework of previous calculations, you can have these settlements recalculated by activating the "**Settlement at user-defined points**" check box.

7.2.12 Calculate user-defined settlements

With user-defined settlements we mean analysis of settlements at any point inside or outside of the foundations. The position of these points is not restricted to the foundation geometry. They can be user-specified in a variety of ways.

If you are completely uninterested in the settlements at the special foundation points, you can go directly to the "**Evaluation**" and "**Special**" menus to calculate and display user-defined settlements.

- "**Evaluation/Settlements at points**" menu item
  With the mouse, you can click on any point within the system. The calculated settlements will then be graphically displayed.
- **Evaluation/Settlements on a line**" menu item
  With the mouse, you click on two points (start and end points of a line). The settlement along this line will then be calculated at constant intervals. If, after calculations are complete, you switch to the "**Evaluation/Automatic multi-node section**" menu item (see Section 9.7.14), you can display the settlement depression for the system section.
- "**Evaluation/Settlements in quadrilateral array**" menu item
  You click on the four points of a quadrilateral. The program then calculates the settlement in a regular array within this quadrilateral. This function is especially useful in connection with the "**Evaluation/Settlement contours**" menu item (see Section 9.7.3), as you have, with settlements calculated in a quadrilateral, a favourable data basis for triangulation and thus for the presentation of contours.

7.2.13 Evaluate and visualise results (Ex. 1)

After calculations are complete the results can be evaluated in a variety of ways.

- "**Evaluation/Settlement location display options**" menu item
  The calculated settlements will be displayed in their appropriate positions. If, after calculating, you have altered input values, the settlements will not be shown. You must then have the system recalculated.
- "**Evaluation/Settlement contours**" menu item
  The program will carry out a triangulation of all calculated settlements. By interpolation within this triangle mesh, which is not identical to the above described triangle mesh for the constrained modulus profile, lines of equal settlement will then be drawn.

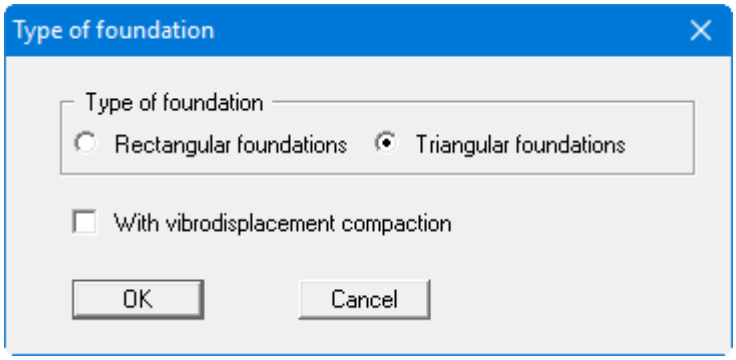
In the "**Special**" menu it is possible to calculate settlement depressions, stress distribution and lines of equal settlement along straight, vertical sections.

In accordance with the principle of ***What you see is what you get*** you can, at any stage of evaluation, send the current screen contents to the printer (see menu item "**File/Print and export**" button "**Printer**", Section 9.1.7).

7.3 Example 2: Analysis of triangular foundations

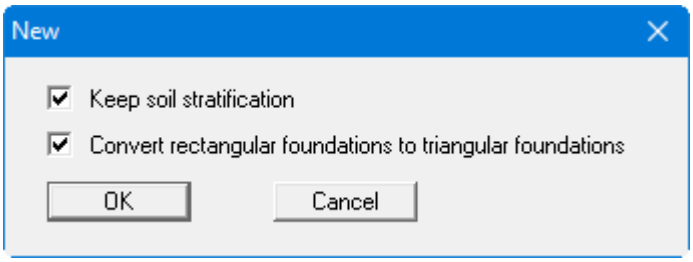
7.3.1 Define borehole points (Ex. 2)

First select the menu item "**File/New**" and activate the "**Triangular foundations**" check box.



To define the drilling points proceed analogous to the descriptions in Sections 7.2.2 to 7.2.5 and enter the coordinates, layer depths and soil properties at the drilling points in the corresponding menu items (see Table 1 to Table 3 in the system data in Section 7.2.1).

If you have previously worked through the example for analysis of rectangular foundations, the following prompt box opens when changing to triangular foundations via "**File/New**":



All of the previous data on the drilling points and the triangular mesh can be retained by activating the "**Keep soil stratification**" check box. The rectangular foundations can also be directly converted to triangular foundations. If the check box is deactivated the foundations must be entered again from scratch.

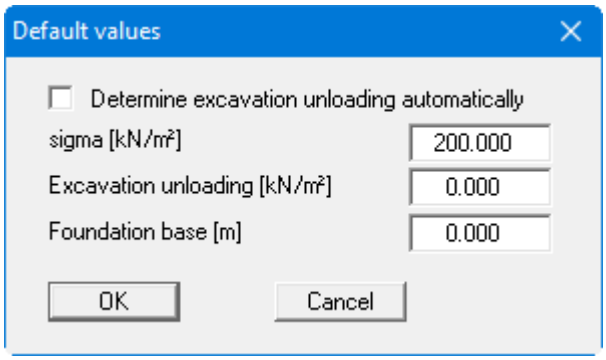
7.3.2 Define triangle mesh (Ex. 2)

The triangle mesh for the constrained modulus profiles is generated analogous to the description in Section 7.2.6. If the "**Keep soil stratification**" check box was activated when changing from rectangular to triangular foundations, the triangle mesh is also transferred.

7.3.3 Define triangular foundations (Ex. 2)

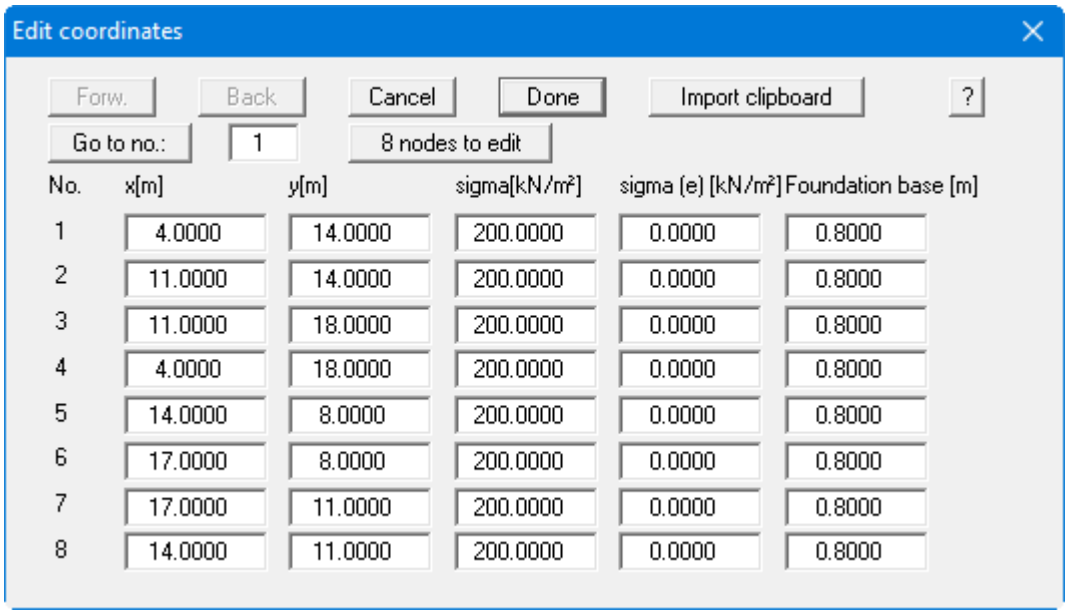
After defining the constrained modulus profile distribution throughout the triangle mesh, you can enter the triangular foundations. The definition of triangular foundations is different to that of rectangular foundations, as a much more flexible input is possible. Input of triangular foundations is very similar to the definition of constrained modulus profiles. If the check box "**Convert rectangular foundations to triangular foundations**" was activated when changing from rectangular foundations to triangular foundations, the foundations' corner points and triangle mesh are transferred directly

If you have started again from scratch, define the corner points of the triangular foundations according to Table 4 in the system data (see Section 7.2.1). You can do this with the mouse, e.g. For this, select the menu item "**Triangular foundations/Define nodes**". Now click on the corner points of the triangular foundation with the mouse. If the same system is to be calculated as in Section 7.2, these are then the four corner points of both foundations. If you define a new triangular foundation corner point with the mouse, this node will be assigned the values given in the menu item "**Triangular foundations/Default values**".



These default values can then be individually adjusted using the menu item "**Triangular foundations/Edit nodes (individually)**".

Alternatively to clicking with the mouse, you can enter the values in tabular form. To do this, select the menu item "**Triangular foundations/Edit nodes (via table)**". The pre-existing data from the conversion from rectangular foundations can be seen in the following dialog box:



Enter the values of the dialog box. You have thus defined 8 nodes of the foundation mesh. Now you must only connect the nodes to a mesh. For this, select the menu item "**Triangular foundations/Manual mesh**" and click on three points of the foundation mesh. The triangular foundation mesh should then look like this:

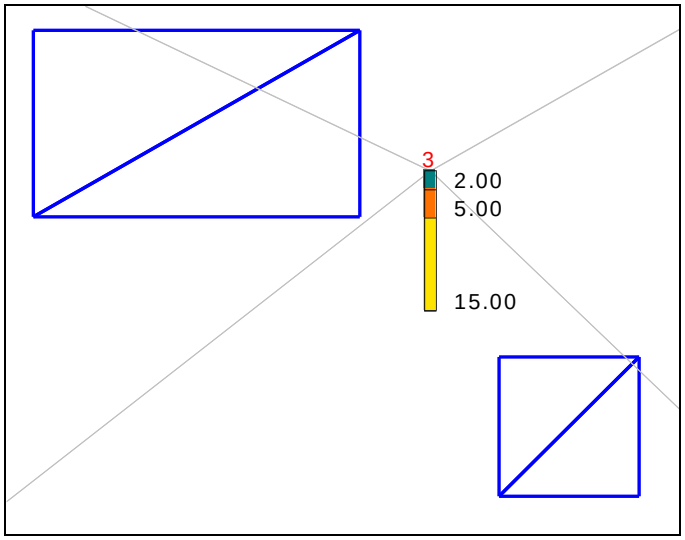
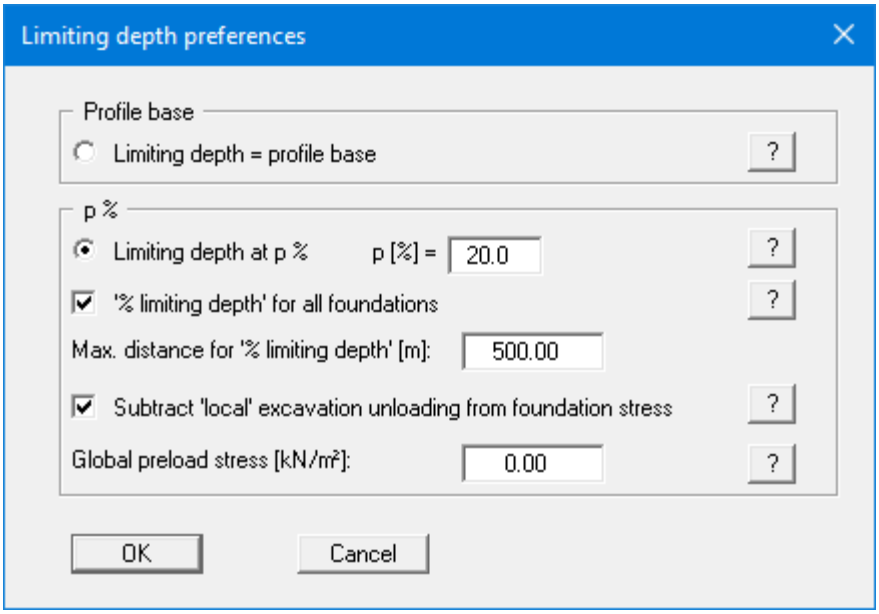


Figure 1 Triangular foundation mesh

7.3.4 Specify limiting depth (Ex. 2)

Calculating triangular foundations the program can determine the limiting depth using two different procedures. Select the menu item "**System/Limiting depth**".



- "**Limiting depth = profile base**"
  The settlements are calculated to the base of the defined soil profile.
- "**Limiting depth at p %**"
  The settlements are calculated to a depth at which the overburden stress * p from the soil weight corresponds to the foundation stress, but not deeper than the profile base. If foundations are close to one another (e.g. with circular foundations), you should, additionally, activate the "**% limiting depth for all foundations**" check box, as otherwise too shallow limiting depths will be calculated. When determining the limiting depth with p %, any excavation unloading for the foundation concerned can be subtracted from the average foundation stress, which usually makes sense. More information on excavation unloading and global preload stress can be had by clicking on the "**Info**" button in the above dialog box. The limiting depth will be calculated in the centre of the rectangular foundation.

If the settlement is to be calculated outside of the foundation concerned, the limiting depth of the foundation which creates the settlement stress is also valid. Stresses above the foundation base generate tensional stresses and will therefore not be considered.

7.3.5 Specify type of settlement analysis (Ex. 2)

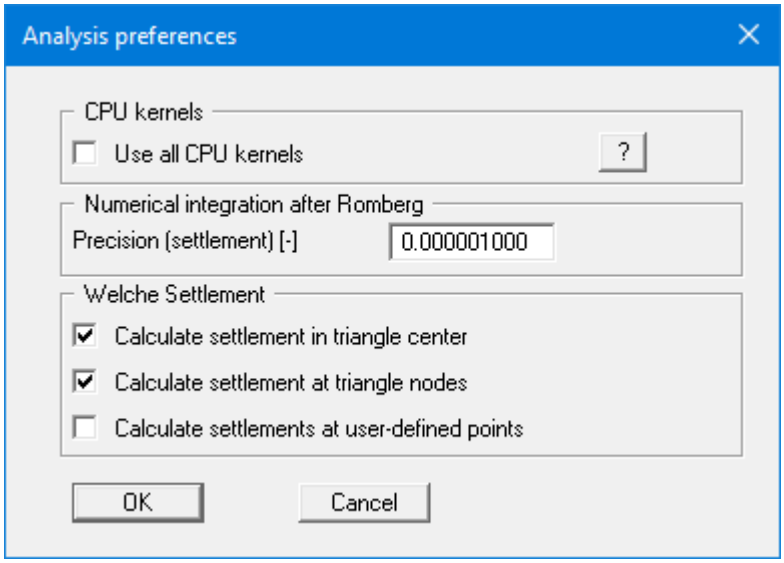
Analogous to the descriptions in Section 7.2.10, the type of settlement analysis is specified using the menu item "**System/Analysis options**".

7.3.6 Calculate foundation settlements (Ex. 2)

After defining a foundation, you can have the system calculated. In principle, we can differentiate between analysis of foundation settlements and the analysis of user-defined settlements. The two calculations can also be mixed. With foundation settlement we mean the settlement at special points on the foundation:

- settlement in the foundation centre
- settlement at the four characteristic points
- settlement at the four foundation corners.

If you are only interested in settlement at the above-described foundations points, select the "**System/Analyse**" menu item.



The analysis can take a lot of time for more complex systems. If you activate the "**Use all CPU kernels**" check box, all CPU kernels of your computer are used. This can considerably reduce the calculation time.

For analysis of triangular foundations numerical integration is required. The integration precision can be specified in the upper area of the dialog box. The default value is sufficiently small for most problems. If the calculated values appear unusual to you, or if you are somewhat cautious by nature, enter a smaller value. This will, of course, cost you calculation time.

Activate the check box(es) of choice and click "**OK**". You may see a note on the specified special preferences first. Acknowledge the notes and start the analysis. Information on analysis progress is displayed in the status bar. Once analysis is complete the foundation settlements are entered in the foundation centre, in the corners and in the characteristic points, depending on the preferences specified above.

If you have calculated a system and then edited any input values (e.g. foundation dimensions, foundation stress, type of limiting depth calculation, etc.), the program deletes all calculated settlements, as they are then no longer valid for the altered system. If you have already defined user-defined settlement points within the framework of previous calculations, you can have these settlements recalculated by activating the "**Calculate settlements at user-defined points**" check box.

7.3.7 Calculate user-defined settlements (Ex. 2)

With user-defined settlements we mean analysis of settlements at any point inside or outside of the foundations. The position of these points is not restricted to the foundation geometry. They can be user-specified in a variety of ways.

If you are completely uninterested in the settlements at the special foundation points, you can go directly to the "**Evaluation**" and "**Special**" menus to calculate and display user-defined settlements.

- "**Evaluation/Settlement at points**" menu item
  With the mouse, you can click on any point within the system. The calculated settlements will then be graphically displayed.
- **Evaluation/Settlements on a line**" menu item
  With the mouse, you click on two points (start and end points of a line). The settlement along this line will then be calculated at constant intervals. If, after calculations are complete, you switch to the "**Evaluation/Automatic multi-node section**" menu item (see Section 9.7.14), you can display the settlement depression for the system section.
- "**Evaluation/Settlements in quadrilateral array**" menu item
  You click on the four points of a quadrilateral. The program then calculates the settlements in a regular array within this quadrilateral. This function is especially useful in connection with the "**Evaluation/Settlement contours**" menu item (see Section 9.7.3), as you have, with settlements calculated in a quadrilateral, a favourable data basis for triangulation and thus for the presentation of contours.

7.3.8 Evaluate and visualise results (Ex. 2)

Once analysis is complete the results can be evaluated in numerous ways. One of these options was previously introduced in Section 7.2.13.

In the "**Special**" menu it is possible to calculate settlement depressions, stress distribution and lines of equal settlement along straight, vertical sections.

In accordance with the principle of ***What you see is what you get*** you can, at any stage of evaluation, send the current screen contents to the printer (see menu item "**File/Print and export**" button "**Printer**", Section 9.1.7).

7.4 Final comparison of rectangular/triangular foundation analysis

A comparison of calculation results for rectangular and triangular foundations will show that there are minor differences in the calculated settlements. This is because of the different calculation point of the limiting depth. With rectangular foundations, the limiting depth is determined in the foundation centre. Triangular foundations do not "know" the centre point of the rectangles. For triangular foundations, the limiting depth is determined at the centre of the triangles, which deviates from the centre of the rectangles. If you select the "**Limiting depth = profile base**" check box (see Section 7.2.9), you will get the same results in both cases, as the limiting depth is then the same for both procedures.

8 Theoretical principles

8.1 General

The program calculates the stresses and strains after the theory of elastic-isotropic half space. Especially in the days when pocket calculators and personal computers were not yet available, comprehensive diagrams and tables were developed. A literature list of tables can be taken from the DIN 4019. Further to this, you are referred to the article "Stress calculation" (Spannungsberechnung) in the "Geotechnical Engineering Pocket Book" (Grundbau-Taschenbuch) (1990; Fourth Edition; also available in english). Here you will also find the complete relationships for stresses and strains below a rectangle in elastic-isotropic half space (formulas 8 to 10 and formulas 14 and 15). These relationships are the basis for the program.

The stress relationships are only used for limiting depth calculations and for the presentation of stress distributions. The strains are calculated directly from the relationships given in the Geotechnical Engineering Pocket Book. A numerical integration with associated loss of precision is therefore not required.

8.2 Characteristic point

The stiffness of foundations cannot be considered using the above mentioned relationships. The foundation loading will be assumed to be a flexible load bundle, whatever the case. It is usual to calculate the settlements at what is called the ***characteristic point***. At this point of the foundation, for a uniform load, the settlements for a flexible load bundle correspond to the settlement of a rigid foundation. The position of the characteristic point is defined as follows:

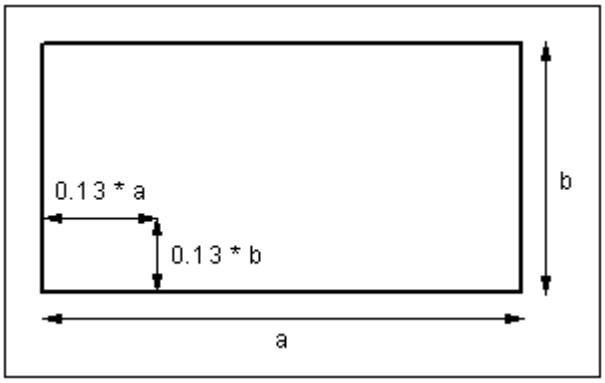


Figure 2 Characteristic point

8.3 Limiting depth

Of great importance for the size of settlements is the limiting depth. The program allows definition of limiting depth in three different ways:

- "**Limiting depth = profile base**"
  The settlements are calculated to the base of the defined soil profile.
- "**Limiting depth = x * b**"
  The settlements are calculated to a depth of x * foundation width, but not deeper than the profile base.
- "**Limiting depth at p %**"
  The settlements are calculated to a depth at which the overburden stress * p from the soil weight corresponds to the foundation stress.

The last possibility is also described in DIN 4019 (see at http://www.din.de). Here, a percentage value p of 20 % is given. This is the program default value. If you have selected this type of limiting depth calculation, the program calculates the overburden stress due to the soil weight for each foundation and compares this value to the average stress due to the foundation load at the characteristic point. The depth distribution of the unit weights for analysis of the overburden stress is determined in the foundation centre. Any excavation unloading for the foundation can be subtracted from the foundation stress, which the usual case.

For settlement analyses, only the stress from the foundation base to the limiting depth will be considered. Thus, for all points outside of the foundation, the limiting depth of the appropriate foundation is valid. Figure 3 shows, in a vertical section, how the settlements of a point, A, are composed if two foundations are defined.

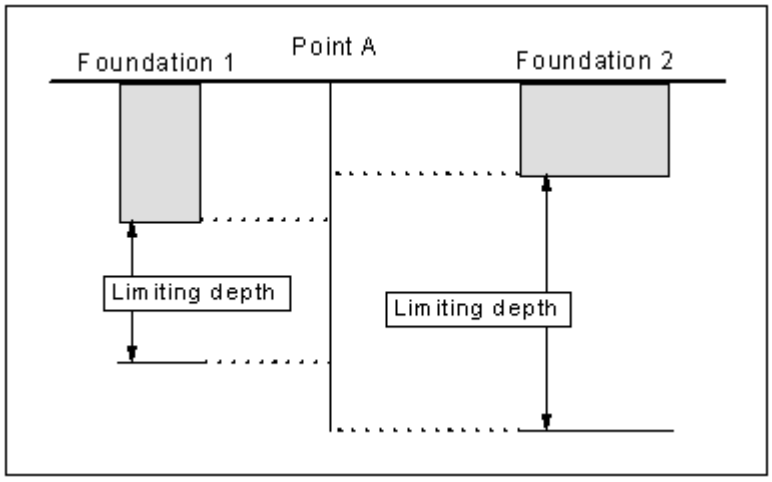


Figure 3 Influence of limiting depth on settlements

If you divide a foundation into several smaller sub-foundations you should, according to theory, get the same settlement values. However, a problem then occurs when calculating limiting depths. The limiting depths of the sub-foundations is smaller due to the smaller widths. Because of the differing limiting depths you will also get different settlement values. To solve this contradiction the **GGU-SETTLE** program offers the possibility of calculating the limiting depth from the stress distribution of all foundations (see Section 7.2.9). If you select this possibility the stress distribution at the characteristic point is calculated from the stresses of all foundations. In this case you will get larger settlements and, depending on the system, more appropriate results.

8.4 Excavation unloading and constrained modulus for reloading

You can define excavation unloading for each foundation. The excavation unloading has the same dimensions as the foundation stress [kN/m²]. It is constant for each foundation. If you have defined an excavation unloading, settlement analyses will be carried out to this value using the constrained modulus for reloading.

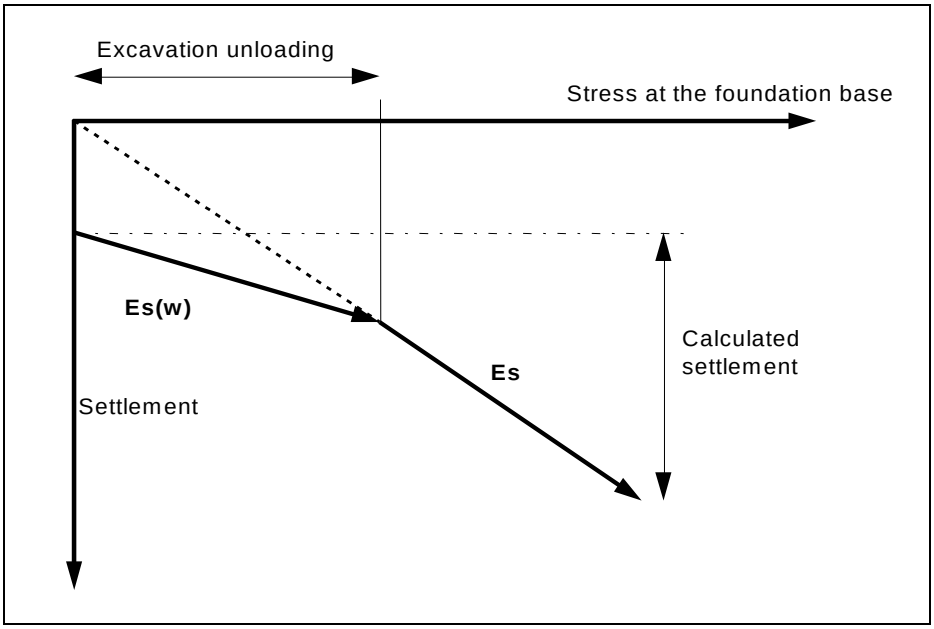


Figure 4 Excavation unloading and constrained modulus

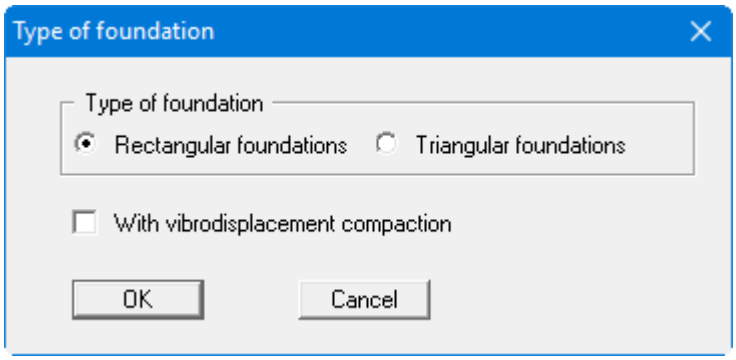
Figure 4 shows the settlements computed for this particular case.

9 Description of menu items

9.1 File menu

9.1.1 "New" menu item

You can enter a new system using this menu item. You will first be asked whether rectangular or triangular foundations are to be generated.



The "**Triangular foundations**" option allows you to calculate settlements and stresses below a flexible foundation of any shape and linear loading. By combining the triangular foundations in mosaic-like manner, any type of load (with reference to ground plan and load size) can be easily modelled. After selecting this menu item the "**Triangular foundations**" menu appears instead of the usual "**Foundations**" menu (see Section 9.5).

Existing soil stratification (constrained modulus profiles) and foundations can be transferred when changing from rectangular to triangular foundations, but when changing from triangular to rectangular foundations only the soil stratification.

Activate vibrodisplacement compaction adoption after Priebe (Heinz J. Priebe, *Die Bemessung von Rüttelstopfverdichtung*, Ground Engineering, December 1995) using the "**With vibrodisplacement compaction**" check box.

9.1.2 "Load" menu item

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

9.1.3 "Save" menu item

You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved without prompting with the name of the current file.

9.1.4 "Save as" menu item

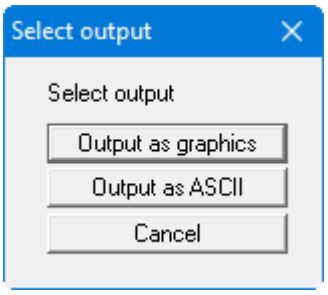
You can save data entered during program use to an existing file or to a new file, i.e. using a new file name. For reasons of clarity, it makes sense to use "**.fda**" as file suffix, as this is the suffix used in the file requester box for the menu item "**File/Load**". If you choose not to enter an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

9.1.5 "Print output table" menu item

9.1.5.1 Selecting the output format

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

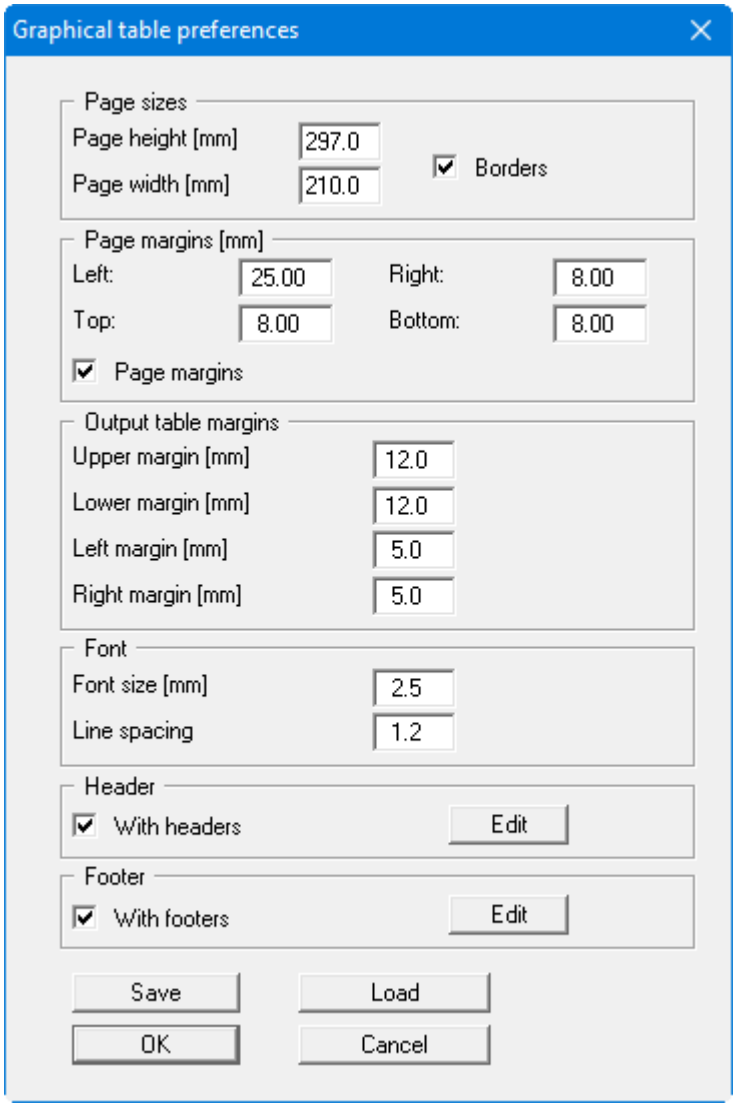
You have the option of designing and printing the output table as an annex to your report within the **GGU-SETTLE** program. To do this, select "**Output as graphics**" from the following options.



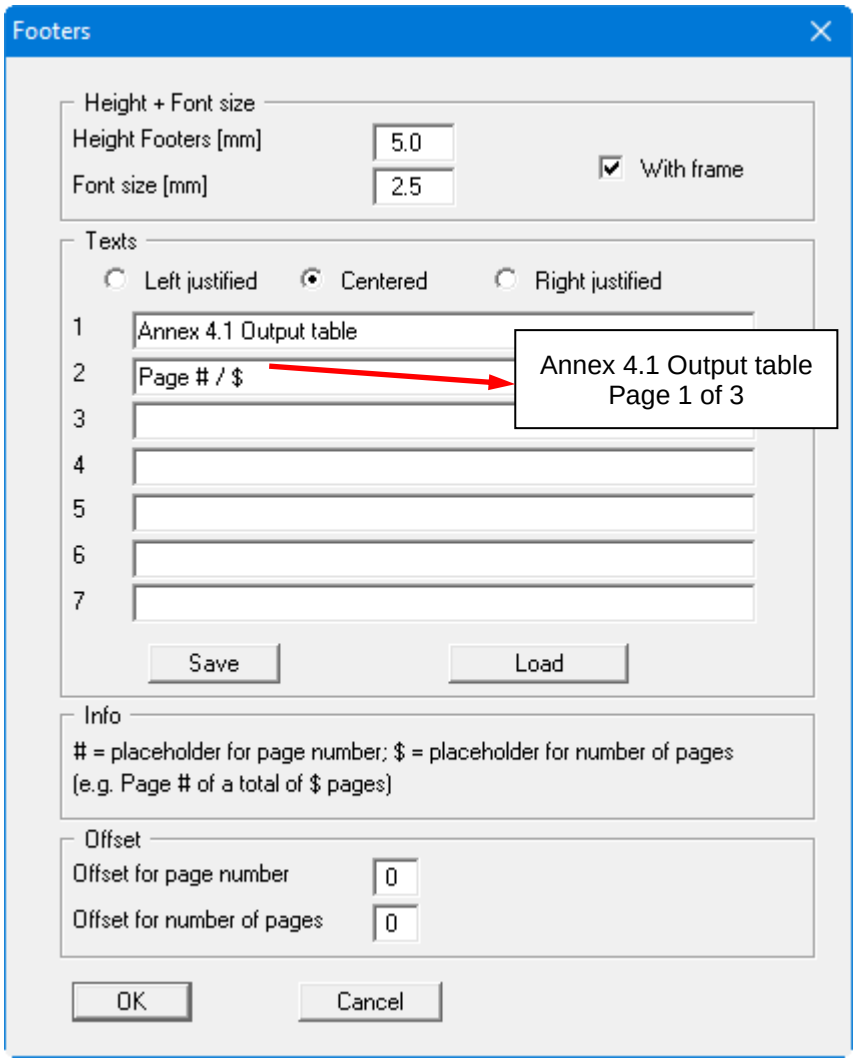
If you prefer to easily print or process the data in a different application, you can send them directly to the printer or save them to a file using the "**Output as ASCII**" command button.

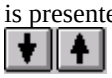
9.1.5.2 Button "Output as graphics"

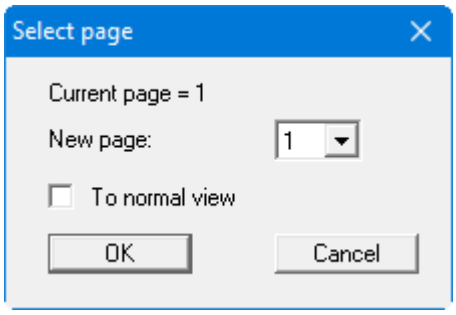
If you selected the "**Output as graphics**" button in the previous dialog box a further dialog box, in which you can define further preferences for result presentation.



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

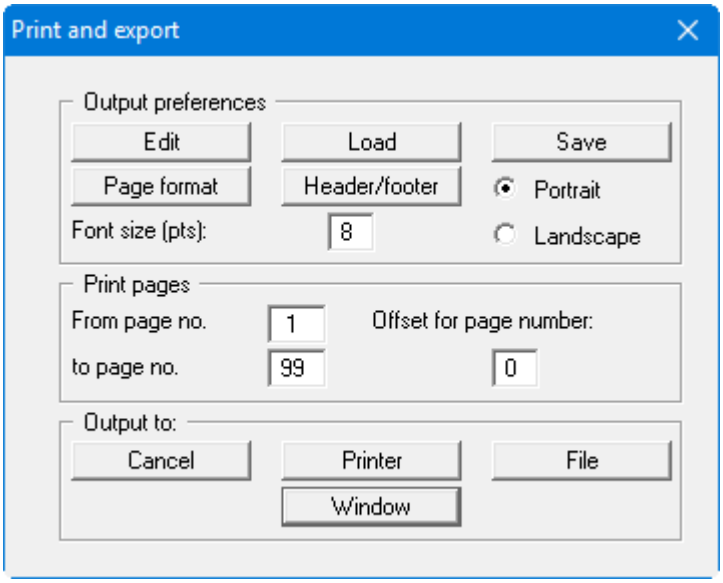


Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "**OK**" you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the "**Start**" button the output table is 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:



9.1.5.3 Button "Output as ASCII"

You can have your analysis data sent to the printer, without further work on the layout, or save it to a file for further processing using a different program, e.g. a word processing application. After selecting the button "**Output as ASCII**" you will see a further dialog box in which you can select the parameters to be used. If you click the "**Start**" button, the following dialog box appears in which you can define output preferences.



In the dialog box you can define output preferences:

- "**Output preferences**" group box
  Using the "**Edit**" button the current output preferences can be changed or a different printer selected. Using the "**Save**" button, all preferences from this dialog box can be saved to a file in order to have them available for a later session. If you select "**GGU-SETTLE.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 "***1***" you can add an ***offset number*** to the check box. This offset will be added to the current page number. The output range is defined using "**From page no.**" "**to page no.**".

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

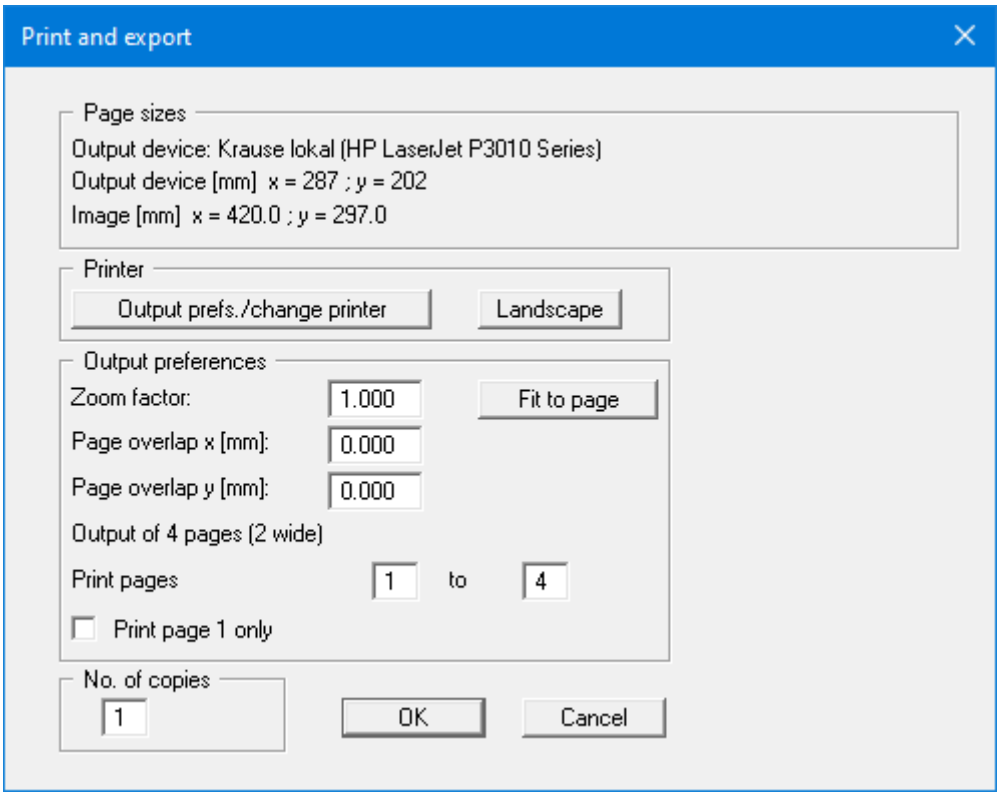
9.1.6 "Output preferences" menu item

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

9.1.7 "Print and export" menu item

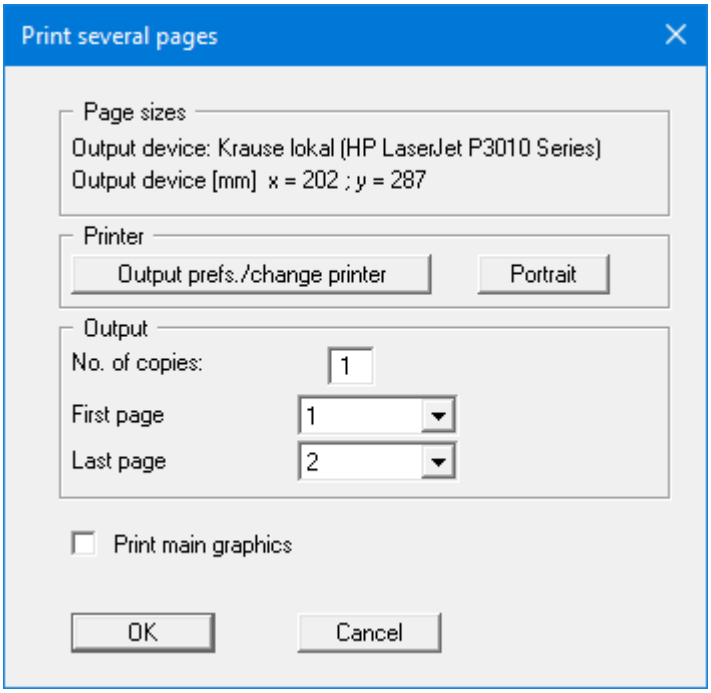
You can select your output format in a dialog box. You have the following possibilities:

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



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

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



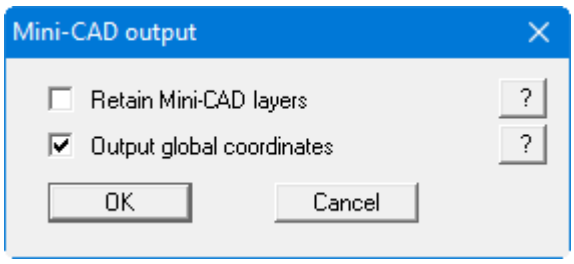
Here, you can select the table pages to be printed. To achieve output with a zoom factor of 1 (button "**Fit in automatically**" is deactivated), you must adjust the page format to suit the size format of the output device. To do this, use the dialog box in "**File/Print output table**" button "**Output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

- "**DXF file**"
  allows output of the graphics to a DXF file. DXF is a common file format for transferring graphics between a variety of applications.
- "**GGU-CAD file**"
  allows output of the graphics to a file, to enable further processing with the **GGU-CAD** program. Compared to output as a DXF file this has the advantage that no loss of colour quality occurs during export.
- "**Clipboard**"
  The graphics are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further processing, e.g. into a word processor. 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**", 6.5).

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

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



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

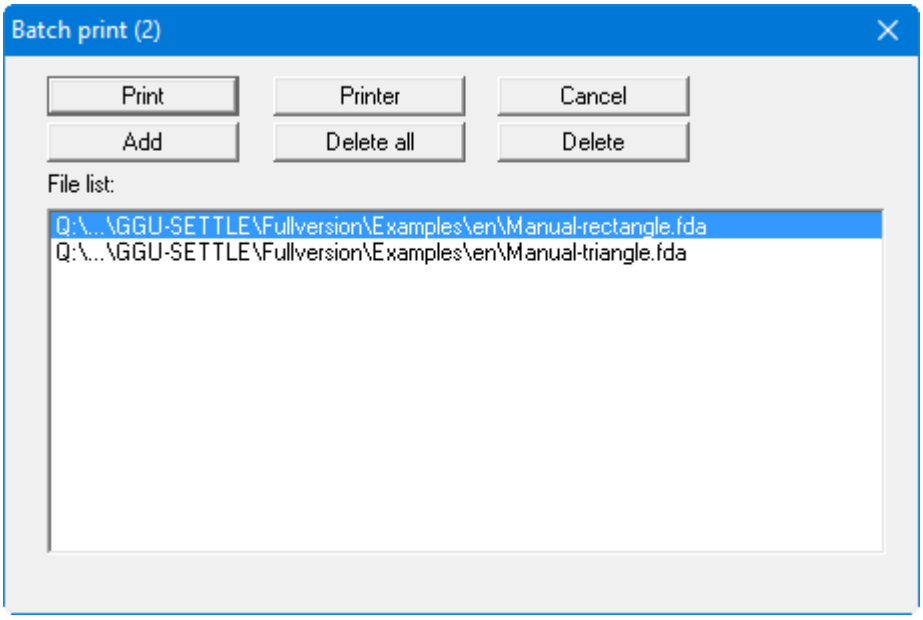
  By activating the "**Output global coordinates**" check box, the present graphics are saved in the system coordinates [m]. Otherwise, they are saved in the page coordinates [mm]. If you import the **Mini-CAD** file saved using "**Global coordinates**" into a different **GGU** program, the coordinates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

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

- "**Cancel**"
  Printing is cancelled.

9.1.8 "Batch print" menu item

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



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

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

9.1.9 "Exit" menu item

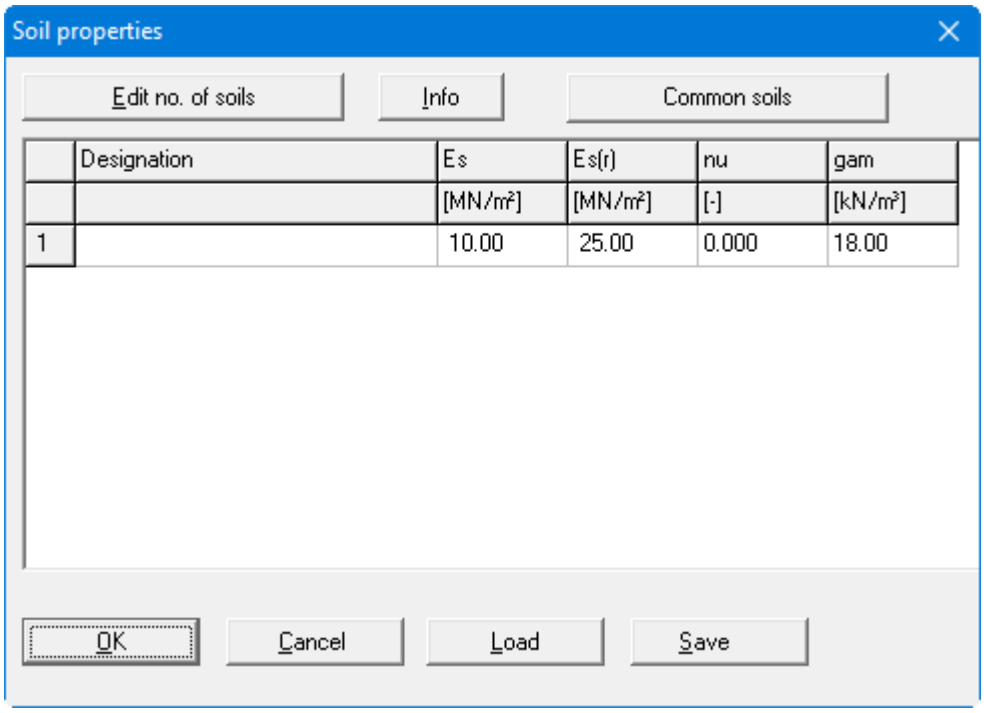
After a confirmation prompt, you can quit the program.

9.1.10 "1, 2, 3, 4" menu items

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

9.2.1 "Layers" menu item

After clicking this menu item, the following dialog box opens for rectangular or triangular foundations. If necessary, edit the number of layers by clicking the "**Edit no. of soils**" button and enter the new number of soil layers.



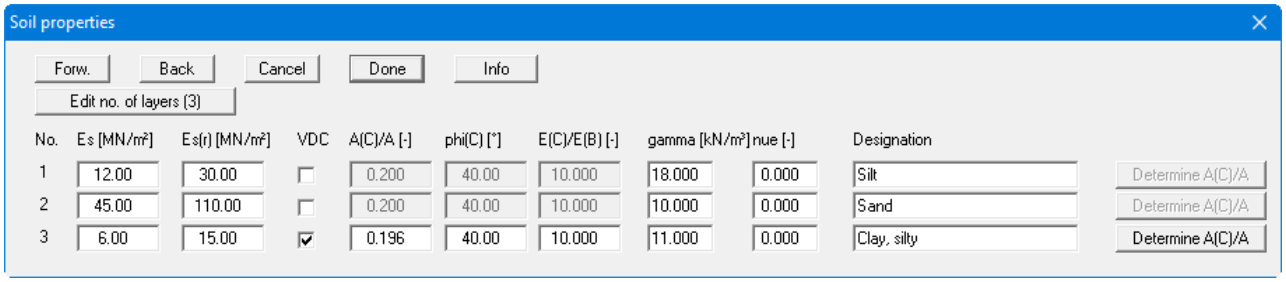
In the above dialog you can enter the name, the constrained modulus for initial loading and for reloading, Poisson's ratio and the unit weights of the individual soil layers. Using the "**Common soils**" button, you can easily select the soil properties of many common soils from a database or determine 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.

In order to ensure a sensible interpolation within the triangle mesh it is necessary that all nodes possess the same number of layers. If your system has areas in which certain soil layers are not present (e.g. peat lenses only in certain areas), then simply assign these layers a thickness of '**0.0**' at the appropriate nodes, using the menu item "**Soils/Default layer depths**".

If nu equals 0.0 for all soils, the value of nu displayed in the ***Soil properties legend*** can then be switched off (see Section 9.9.9)

The current stratification with all corresponding data can be saved to a separate file using the "**Save**" button in order to have it available for a different system, without a lot of typing work, using the "**Load**" button.

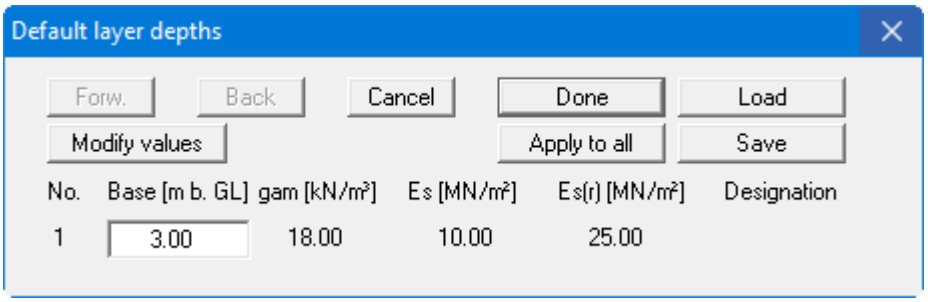
If you are working with vibrodisplacement compaction the following dialog box opens after clicking this menu item:



Define the soil layer improved by vibrodisplacement compaction by activating the "**VDC**" check box. Enter the corresponding parameters for this layer (see "**Info**" button). The ratio of the areas "**A(C)/A**" can be determined and adopted in the table by pressing the far right button. The ratio "**E(C)/E(B)**" should not be greater than 15 to a maximum of 20.

9.2.2 "Default layer depths" menu item

After clicking on this menu item the following dialog box appears:



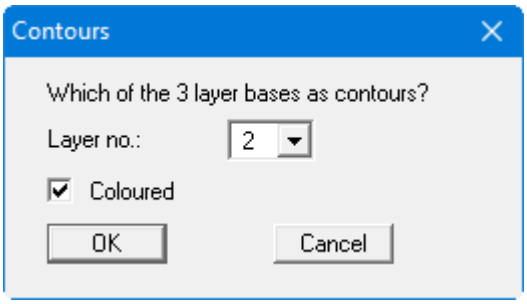
If you have defined three layers using the "**Soils/Layers**" menu item, three layers will also be shown in this dialog box. The corresponding constrained moduli are also shown for information purposes. The given depths can be edited to suit your wishes. Depths are entered positive downwards as m below ground level (= 0.0 m). If you have activated the use of absolute heights in the "**System/Ground level**" menu item (see Section 9.6.3), enter the depths as absolute elevations here, for example in m AD.

During the following definition of triangular nodes, these layer depths are assigned to the appropriate nodes. The layer depths for each node can be edited as wished. If the constrained modulus profile is equal for almost all nodes, you can save input work during the initial definition of default depths.

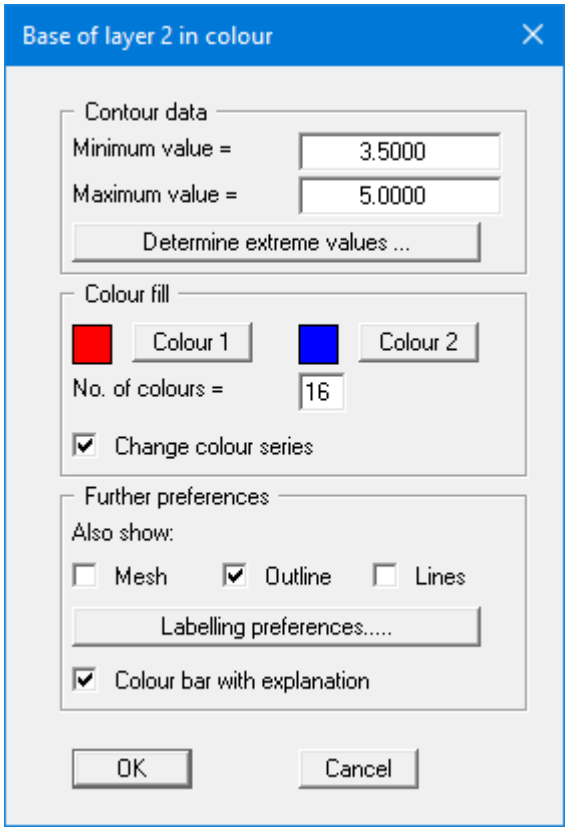
Using the "**Load**" button you can load a previously saved depth distribution. Using "**Save**" you can save the current depth distribution to a file, in order to have them available later, without renewed input. Using the "**Modify**" button, all existing depths can be increased or reduced by applying a constant. If you select the "**Apply to all**" button, all current triangle nodes will be assigned these depths. Using the "**Cancel**" button, you may leave the dialog box without accepting the alterations. You can also leave the dialog box with "**Done**". Any changes will be accepted.

9.2.3 "Layer base contours" menu item

This menu item serves as a check for the layer depth input data. The following dialogue box appears where you enter the number of the layer for which you would like to see a contour diagram of the base.



If you activate the "**Coloured**" check box you will see a colour filled contour diagram. Otherwise a normal contour diagram will be drawn (also see Section 9.7.3.2).



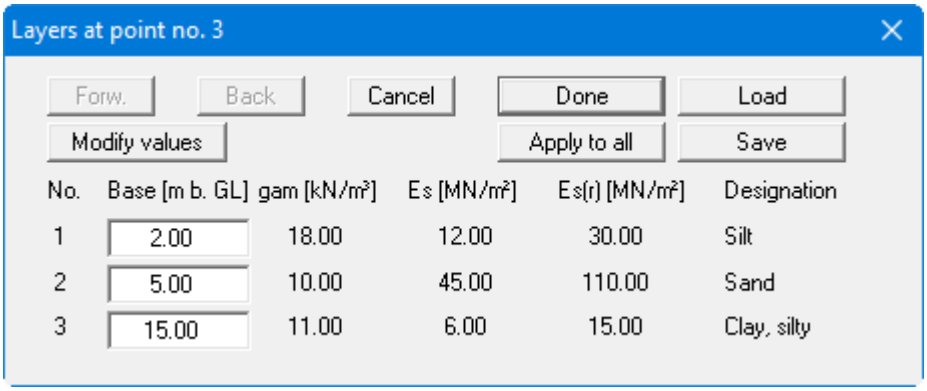
For colour-filled graphics you will see the above dialog box.

- "**Contour data**" group box
  Press the "**Determine extreme values**" button. The program then determines the minimum and maximum values for the corresponding layer base. You can then edit these values, for example in order achieve a defined start value.

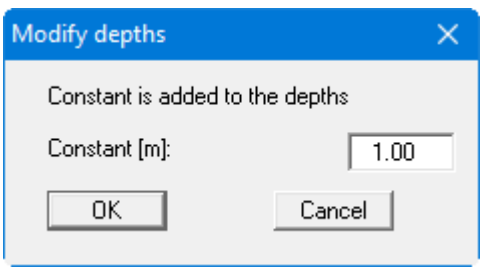
- "**Colour fill**" group box
  You can control the colour subdivisions of the contour diagram using "**No. of colours**". In the example above, 16 colours will be displayed between "**Colour 1**" and "**Colour 2**". The default setting is a colour course from red to blue. These colours can be edited as required after selecting the "**Colour 1**" and "**Colour 2**" buttons, or simply reverse the choice by selecting the "**Change colour series**" check box.
- "**Further preferences**" group box
  In addition to the colour presentation you can also have the triangle mesh and/or the outline displayed. Additional contour lines can also be drawn. Line labelling preferences can be defined by means of the "**Labelling preferences**" button. Here you can also define the font size for the colour bar on the right-hand edge of the screen, which is used to assign the respective colour to the corresponding size on your output sheet. Activate the check box for labelling the colour bar with the explanation of the value displayed.
- "**OK**"
  The colours will be drawn after confirmation. If this colour bar is drawn in the right page margin, specify a larger value for the right plotting margin (e.g. 25 mm) in the "**Page size + margins/Page size and margins**" menu item (see Section 9.10.5).

9.2.4 "Edit layer depths" menu item

This menu item allows editing of the base of a layer at the individual nodes of the triangle mesh. After clicking the menu item or activation using [**F3**] a message box opens. After double-clicking near the node to be edited you can change the depth of the layer base.

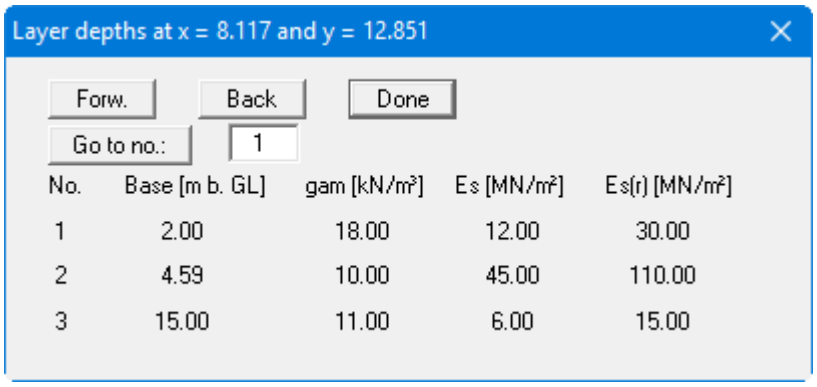


Using the "Modify" button, all existing depths can be increased or reduced by applying a constant. For example, if all depths are to be displaced by one metre, you only need to enter the constant once and all depth data for this profile will be adapted.



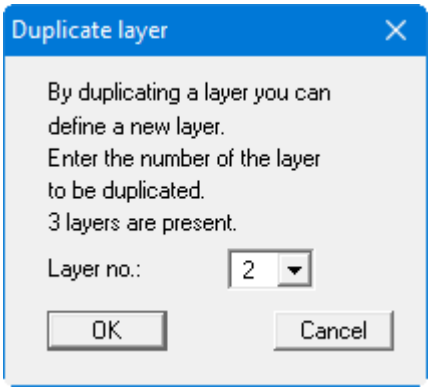
9.2.5 "Determine layer depths" menu item

Have the constrained modulus profile displayed for any point within the triangle mesh. Click the required point after selecting this menu item or activation via [**F7**]. The results for the selected point are then displayed in a dialog box.



9.2.6 "Duplicate layer" menu item

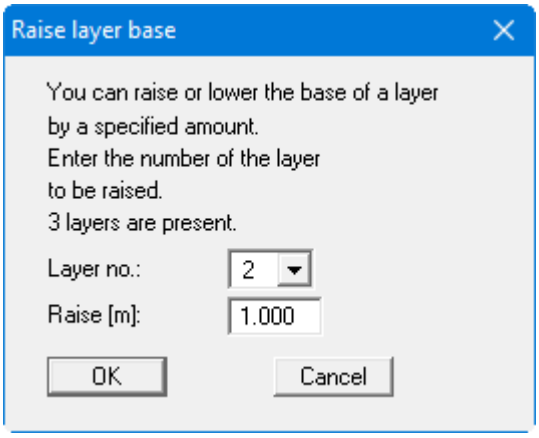
Sometimes it is necessary to duplicate a layer into an existing system. Using the previously described menu items this is strenuous.



A new layer can be defined by partitioning an existing layer. The depth range of the selected layer is divided into 2 new layers with the same depth. The final depth of the constrained modulus profile thus remains neutral. The soil properties of the selected layer are adopted for both new sublayers.

9.2.7 "Raise layer base" menu item

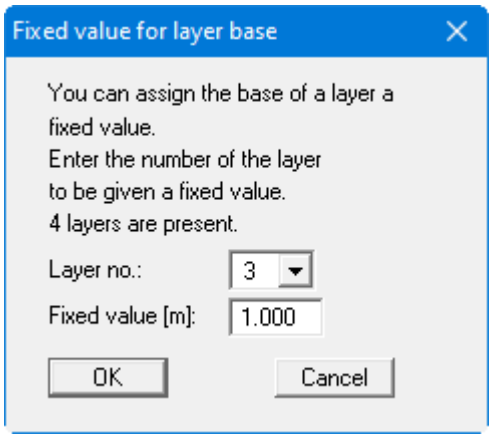
Sometimes it is necessary to raise or lower the base of layer in an existing system by a certain amount. Using this menu items the following dialog box appears:



A layer can be selected and raised or lowered by a given amount. Lowering of a layer is achieved using negative values. This action applies to all constrained modulus profiles. If the layer above or below are incompatible, the rise will limited to the necessary value.

9.2.8 "Fixed layer base" menu item

With this menu item a selected layer in all constrained modulus profiles can be retroactively assigned a fixed value as layer base.

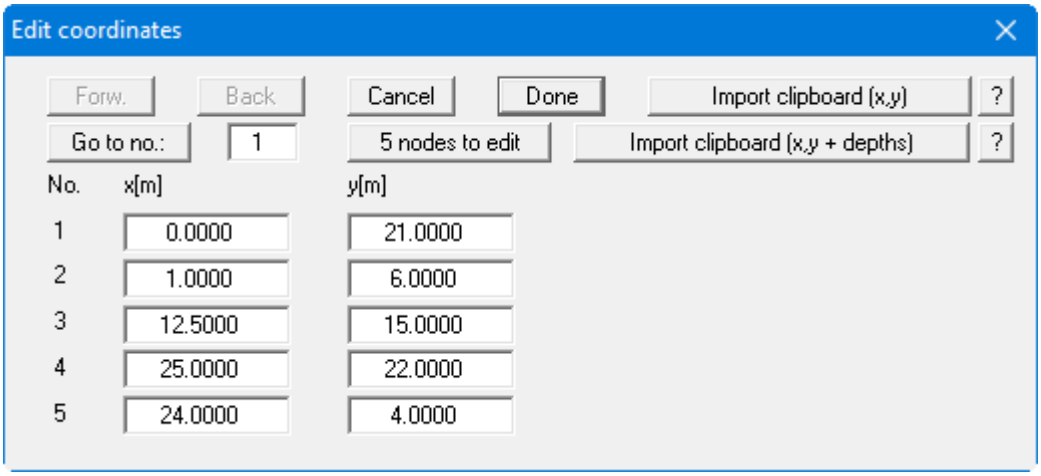


9.3.1 "Define nodes" menu item

Use the left mouse button to define a new node or the right mouse button to delete a previously defined node. Each new constrained modulus profile is assigned the default depths (see menu item "**Soils/Default layer depths**", Section 9.2.2). If a constrained modulus mesh has already been generated using existing nodes, new nodes can only be generated outside of the existing mesh.

9.3.2 "Edit nodes (via table)" menu item

Using this menu item you can edit the coordinates of the existing nodes or enter new ones. After clicking on it the following dialog box appears:



If the current number of nodes needs to be edited, click the "**x nodes to edit**" button and enter the new number of nodes. It is also possible to delete nodes in this way . Navigate through the table using "**Forw.**" and "**Back**". Each new constrained modulus profile is assigned the default depths defined in menu item "**Soils/Default layer depths**" (see Section 9.2.2).

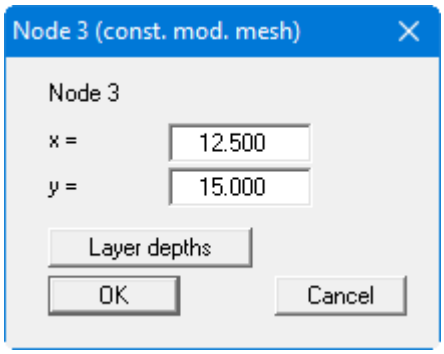
If you already have the coordinates and/or layer depths available in a text editor or in Excel, they can be imported from the Windows clipboard using the two corresponding buttons. Information on this is provided via the two "**?**" buttons. By pressing the lower button, "**Import clipboard (x,y + depths)**", the program anticipates 2 columns for x and y and then as many columns as you have defined soil strata.

9.3.3 "Move nodes" menu item

After clicking this menu item an info box is displayed. The nodes (constrained modulus profile) can be moved when holding the left mouse button. The coordinates of the current node are displayed in the title bar. The last node movement can be undone using the [**Backspace**] key.

9.3.4 "Edit nodes (individually)" menu item

Using this menu item it is possible to retroactively edit individual nodes of the constrained modulus mesh. By double-clicking near a node the following dialog box opens, allowing the coordinates to be edited.



After clicking the "**Layer depths**" button in the above dialog box or by clicking directly in the constrained modulus profile, the layer depth dialog box opens; it can also be opened via the menu item "**Soils/Edit layer depths**" (see Section 9.2.4).

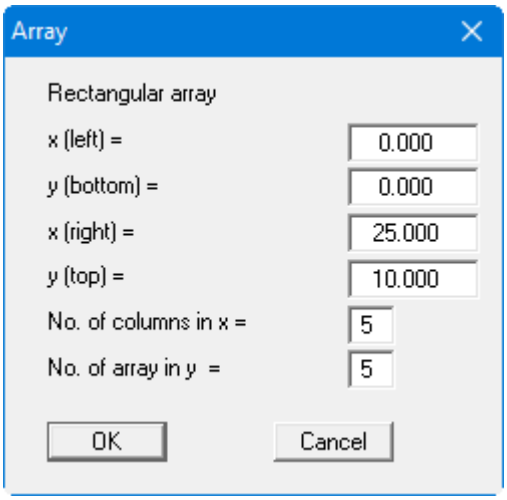
9.3.5 "Generate nodes in array" menu item

The nodes can be defined on an array using this menu item: A dialog box opens for specifying the type of array:

- "**Line**"- along one or more lines,
- "**Rectangle**" - in one or more rectangles,
- "**Quadrilateral**" - in one or more quadrilaterals.

The procedure is similar for all three cases. Therefore, only the rectangles will be described.

"**Rectangle**"



Enter the corner points of the array and the number of subdivisions. If one of the node points thus generated were to lie within an existing mesh, node generation is aborted with an error message and no further nodes are generated.

If an array is defined using a line the coordinates of the two end points are entered and those of the four corners for a quadrilateral array.

9.3.6 "Manual mesh" menu item

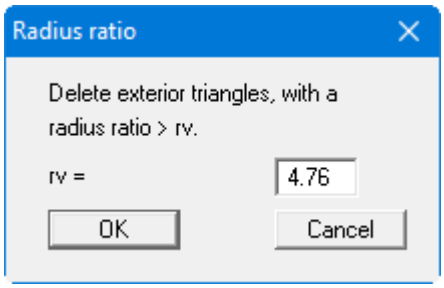
After input of the nodes this menu item is used to define the mesh. Three nodes must be clicked using the left mouse button. An triangle element can be deleted by selecting the three corresponding nodes once again using the left mouse button.

9.3.7 "Automatic" menu item

After entering the mesh nodes automatic mesh generation can be carried out using this menu item (Delauney triangulation). If a triangle mesh already exists it can be either deleted or supplemented. Only in exceptional cases you should select the "**Supplement**" button, as triangulation follows certain laws which may not allow sensible complementing of a current partial mesh.

9.3.8 "Round off" menu item

During Delauney triangulation a triangle mesh is generated that envelops all nodes. This can lead to acute-angled triangle elements in the boundary regions. These triangles can be removed from the triangle mesh using this menu item.



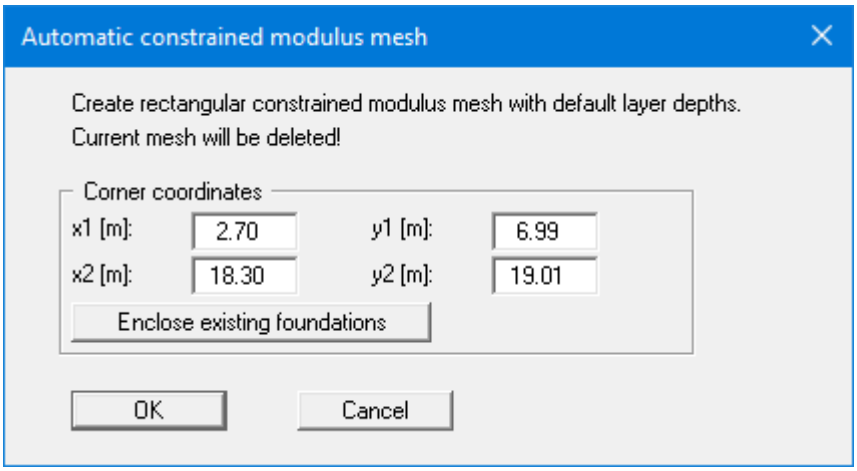
The radius ratio describes the relationship between external radius and internal radius of a triangle. For an equilateral triangle, this ratio equals 2.0 (optimum). Before you arrive at the dialog box above you will be shown the maximum and minimum radius ratios. In the example above, all external triangles with a radius ratio greater than 6 will be removed. To avoid chaos, only exterior triangles will be removed.

9.3.9 "Delete nodes and mesh" menu item

With this menu item you can delete selected system triangles. You must first click four points in anti-clockwise direction. All triangles with their centroid within this quadrilateral will be deleted.If you want to delete the entire network but not the nodes, bypass the entire area. If the "**Delete all nodes and mesh**" button is pressed, the complete mesh and all nodes are deleted.

9.3.10 "Automatic constrained modulus mesh" menu item

With this it is possible to quickly generate a rectangular constrained modulus mesh with default depths for further processing. Both the nodes are set, and the nodes are networked. The following dialog box opens for entering the corner coordinates of a rectangular mesh.

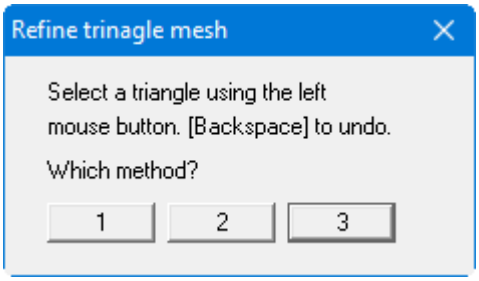


If the "**Include all foundations**" button is clicked the program automatically searches for the most suitable coordinates and enters them in the dialog box as shown above.

9.3.11 "Refine individual elements" menu item

The program allows the refinement of a current triangle mesh. Refinement is generally only then necessary when you would like to supplement a current triangle mesh with further, possibly fictitious, nodes.

Using this menu item, it is possible to select individual elements of the constrained modulus mesh for refinement, after selecting the refining method:



Three different methods can be used for triangle mesh refinement. Method '**3**' generally gives the best mesh refinement and is recommended.

Three different refinement methods can be applied for element refinement. Refinement will be demonstrated on the following mesh using element 23 as an example.

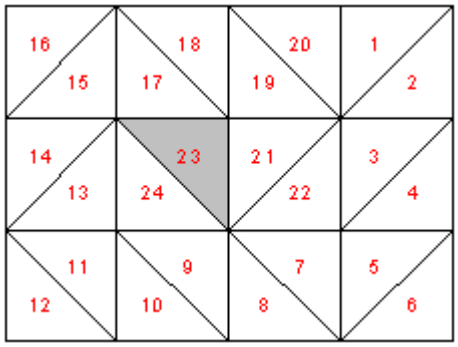


Figure 5 Refinement demonstration mesh

- "**Method 1**"
  An additional node is generated in the centroid of the selected triangle.

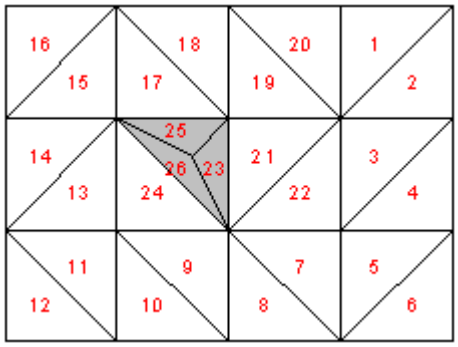


Figure 6 FEM mesh refinement using Method 1

- "**Method 2**"
  The selected triangle element and the neighbouring triangle element are halved.

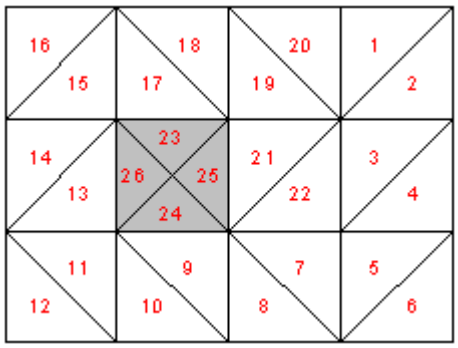


Figure 7 FEM mesh refinement using Method 2

- "**Method 3**"
  A new triangle element is inserted at the median of the clicked triangle element. The neighbouring triangle elements are halved.

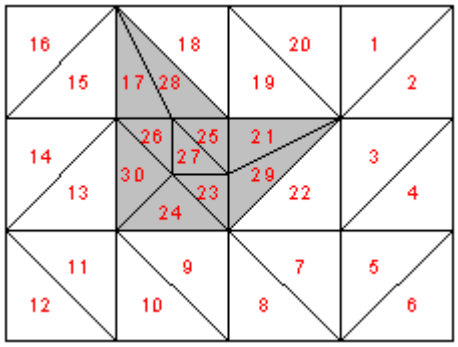


Figure 8 FEM mesh refinement using Method 3

9.3.12 "Refine elements in section" menu item

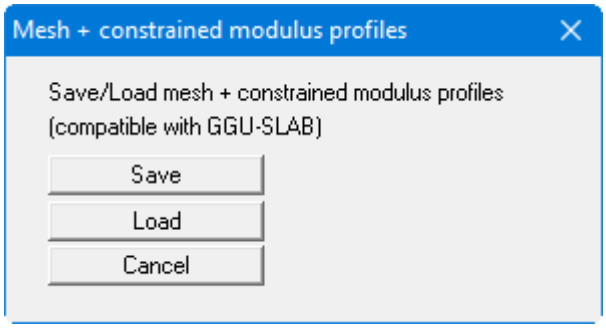
A number of elements previously enveloped in a polygon can be refined using this menu item. A description of the 3 refinement methods can be found in Section 9.3.11.

9.3.13 "Refine all elements" menu item

All elements of the triangle mesh can be refined using this menu item. A description of the 3 refinement methods can be found in Section 9.3.11.

9.3.14 "Save/Load" menu item

The nodes and triangle elements of the constrained modulus mesh can be saved or loaded. The following dialog box opens:

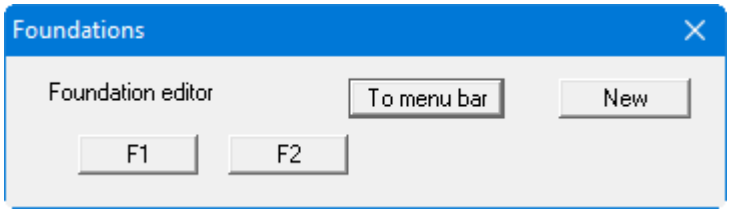


Using files with the "**.ggu_stei**" extension it is possible to open constrained modulus meshes created using the **GGU-SLAB** analysis program in **GGU-SETTLE** or, in the opposite direction, to export from **GGU-SETTLE** to **GGU-SLAB**.

9.4.1 "Define" menu item

9.4.1.1 Create/select foundations

If you are working with rectangular foundations, you will see the following dialog box for inputting new or editing existing foundations using this menu item:

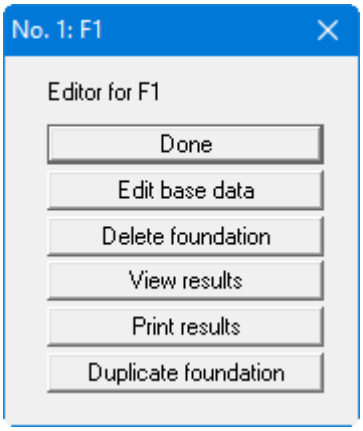


Three foundations are already present in the dialog box. The entered foundation designations are shown on the respective buttons. The following actions are possible:

- "**To menu bar**"
  You return to the original menu bar.
- "**New**"
  You can now enter data for a new foundation.
- "**F1**", "**F2**"
  By clicking the buttons labelled with the foundation designation you can open and edit the data for the corresponding foundation.

9.4.1.2 Operations on selected foundation

After clicking on "**New**" or on the button of an existing foundation, the following dialog box opens:

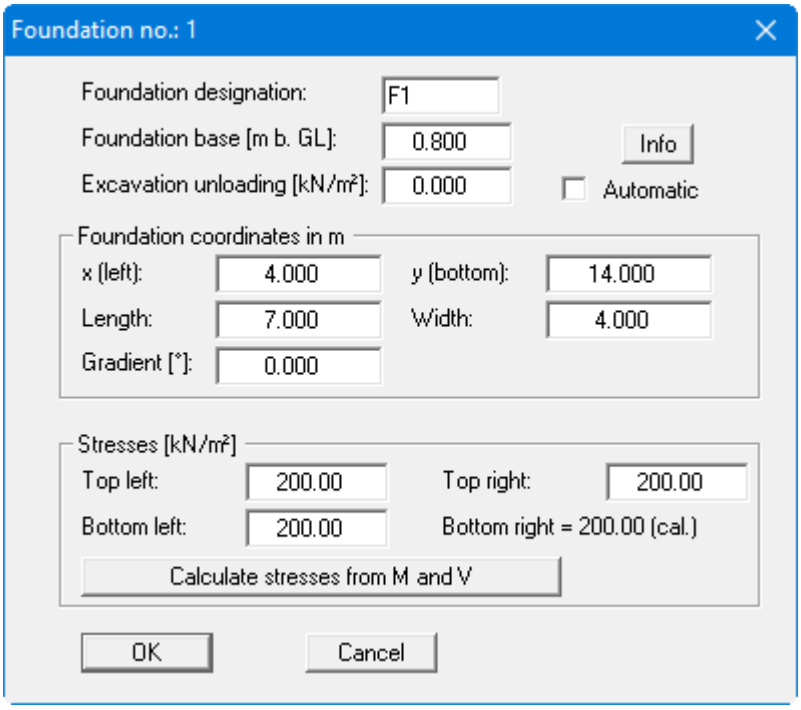


The following operations are possible:

- "**Done**"
  You return to the original menu bar.
- "**Edit base data**"
  You can enter or edit the base data for the corresponding foundation (see the following Section 9.4.1.3).
- "**Delete foundation**"
  After a safety request, the current foundation can be deleted.
- "**View results**"
  The calculated settlements will be displayed. If certain settlement values have not yet been calculated, the corresponding entries will remain empty.
- "**Print results**"
  The results for the currently selected foundation will be sent to the printer.
- "**Duplicate foundation**"
  You can duplicate the current foundation. You can then edit the base data of the duplicated foundation.

9.4.1.3 Edit base data

After clicking in the "**Edit base data**" box, you can enter or edit the base data for the corresponding foundation. The following window will open:



You enter foundation designation and foundation base. Furthermore, an excavation unloading can be entered. If the value excavation unloading > 0.0, the constrained moduli for reloading will be used in settlement analyses, for the proportion of the excavation unloading which is "re-used". This corresponds to the reloading case of a time-settlement diagram. For the load component over and above this, or for a system without excavation unloading, the constrained modulus for initial loading will be used.

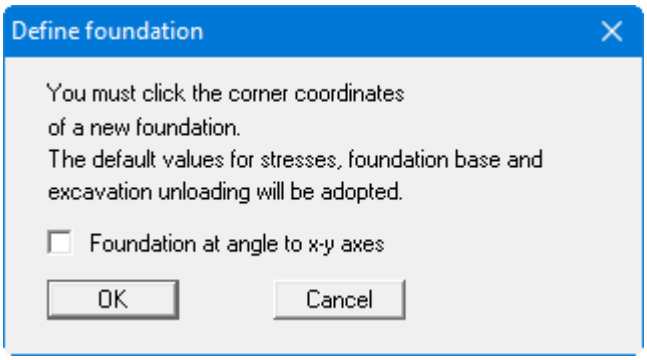
In the "**Foundation coordinates in m**" group box you enter the x and y ordinates and the length and width of the foundation. Further to this, an inclination to the horizontal can be given so that non-axis-parallel foundations can also be analysed.

In the "**Stresses [kN/m²]**" group box you can enter the corner stresses of the foundation. As the stress distribution within a foundation is assumed to be linear, three corner stresses are sufficient to describe the stress distribution. The fourth is calculated internally by the program. It will be visible in the window when it opened the next time. By clicking the "**Calculate stresses via M and V**" button, you can have the foundation stresses calculated from the moment and vertical force (if these values are known).

After clicking the "**OK**" button, input or alterations will be accepted; clicking on "**Cancel**" you leave the box without accepting your alterations.

9.4.2 "Define graphically" menu item

This menu item allows the input of rectangular foundations using the mouse.

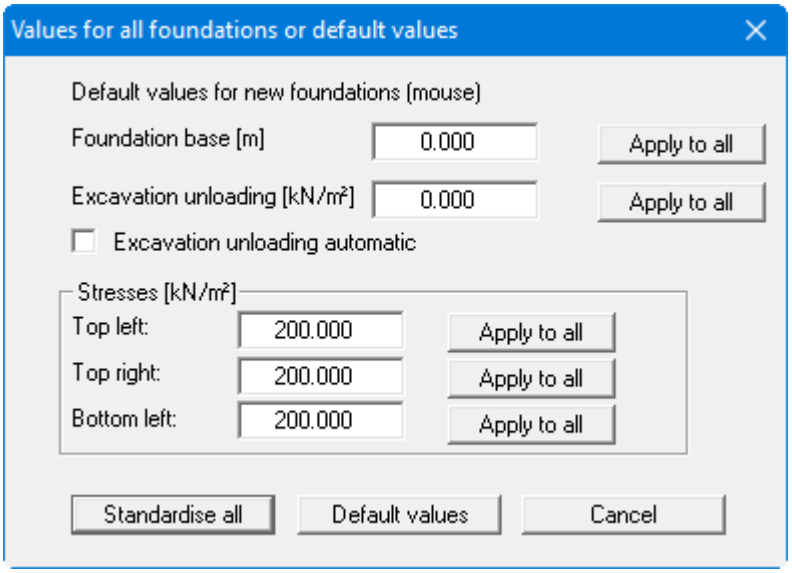


After confirming with "**OK**" you click on the two corners of a foundation with the left mouse button. If the "**Foundation at angle to x-y axes**" check box is activated an ***inclined*** foundation (inclined to the horizontal) can be entered with mouse support. In this case you must click on three points with the mouse. After defining the points, the dialog box for entering the basic data opens, allowing a foundation designation to be entered, among other things (see the input explanations in Section 9.4.1.3).

You can greatly simplify input if you have a scanned foundation template. You can integrate such graphics using the **Mini-CAD** system. The scaling of the graphics is a problem which is easy to overcome. If you know the separation of two points in the template then simply draw a line with **Mini-CAD**, which connects these two points. Then double-click on this line. In the dialog box which then appears, the length of the line is given. From this length and the known length of the line, you can determine the scale discrepancy. Then select the "**Modify object**" icon from **Mini-CAD** and enter the scale distortion. Voila! The imported graphics are now true to scale. You now need only click on the foundation corner points. You can then delete the scanned graphics from the presentation.

9.4.3 "Values for all foundations" menu item

Using this menu item, it is possible to specify certain foundation data globally for all foundations. After clicking on it the following dialog box appears:



The following input data can be globally specified:

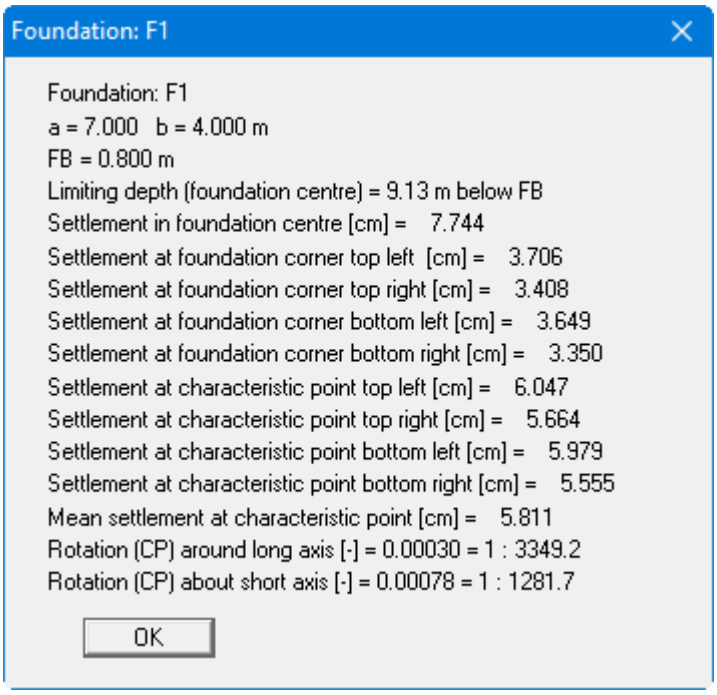
- foundations base,
- excavation unloading,
- corner stresses of the foundations.

An "**Apply to all**" button follows each input box. By clicking any button, the respective data is adopted for all foundations. You then return to the above dialog box and can edit further data or exit the box via the "**Cancel**" button. The "**Apply to all**" button adopts all data for all foundations. The dialog box is closed automatically after this button is clicked.

The values entered above are saved as default values by pressing the "**Default values**" button. These default values are adopted as the basic data for graphically generated foundations (see Section 9.4.2) and for foundations generated by the program (see Sections 9.4.9 to 9.4.11). Subsequent changes to the data can be made at any time via the menu item "**Foundations/Define**" (see Section 9.4.1).

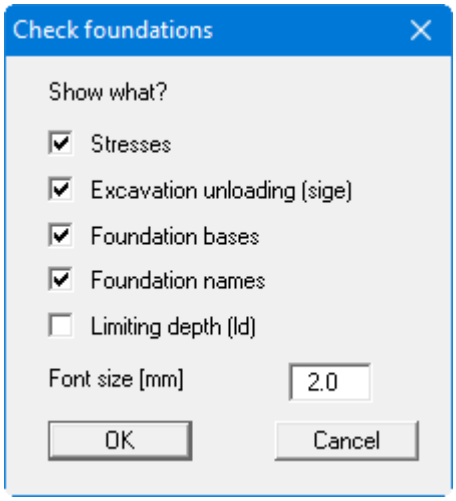
9.4.4 "Results" menu item

The results of the settlement analysis for selected foundations can be viewed in a message box using this menu item. If the system has not yet been analysed, only the foundation dimensions and the limiting depth are displayed in the message box. For example, after clicking an analysed foundation the following results are displayed:



9.4.5 "Check foundations" menu item

The foundation input data can be checked via this menu item or, alternatively, by pressing [**F8**]. The individual elements to be visualised can be specified using the check boxes in the dialog box.



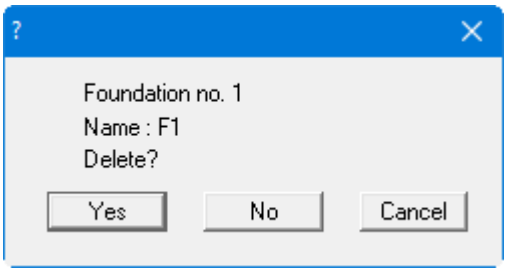
Adopt the preferences by pressing "**OK**". The selected data are then displayed in the foundations visualised on the screen. The abbreviations used are described in the ***General legend*** (see Section 9.9.8).

9.4.6 "Stress contours" menu item

The foundation stress distributions can be visualised as a coloured contour diagram with the aid of this menu item. The procedure is in complete analogy to the visualisation of layer base contours (see Section 9.2.3).

9.4.7 "Delete individual foundations" menu item

With this menu item individual foundations can be deleted. The program will go through all foundations in the order in which they were entered. For each foundation the following dialog box appears:



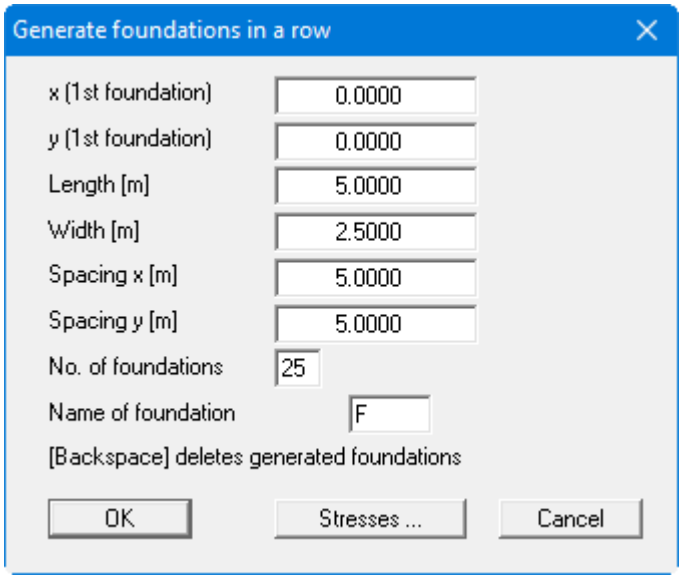
If you do not want to delete the foundation shown, click "**No**". If there are a number of foundations, the box for next foundation opens.

9.4.8 "Delete all foundations" menu item

After a safety prompt, you can delete all current foundations.

9.4.9 "Generate foundations in a row" menu item

You can generate several foundations of equal dimensions in a row. The following dialog box opens:



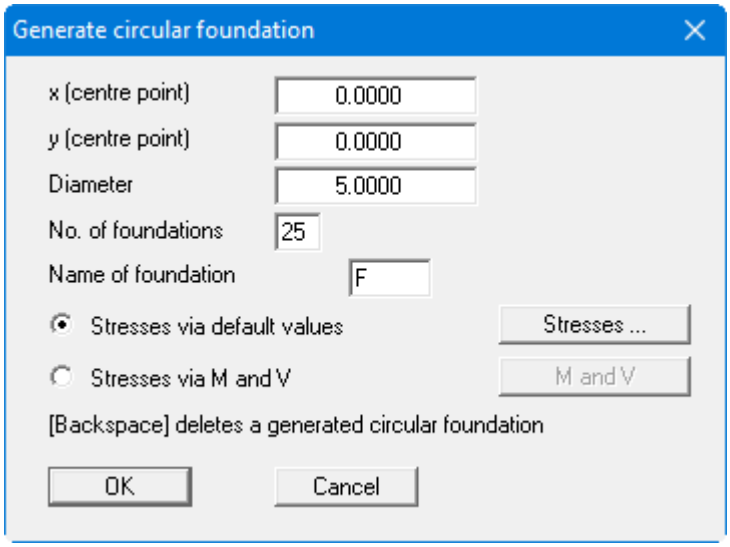
First, enter the x and y ordinates of the first foundation. Define the length and width of all foundations. With "**Spacing x**" and "**Spacing y**" the displacement of the generated foundations to each other is given. The name of the foundation is supplemented with the current number (start number = 1).

The foundation base, the excavation unloading and the foundation stresses will be entered in accordance with the default values in the "**Foundation/Values for all foundations**" menu item. The default values can be adapted via the "**Stresses …**" button.

Immediately after generation, the created foundations can be deleted by pressing the [**Backspace**] key, if the foundation generation is not according to your wishes.

9.4.10 "Generate circular foundation" menu item

Using this menu item you can combine several foundations to a circular foundation.



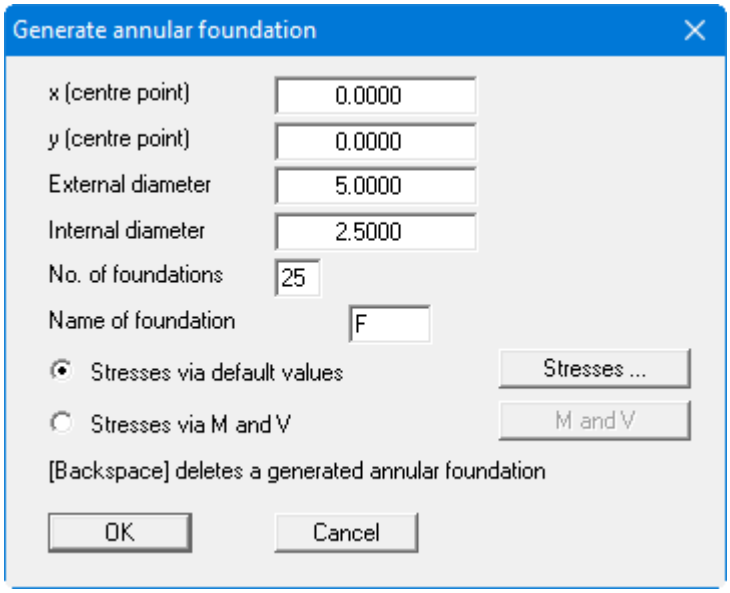
First, enter the x and y ordinates of the centre point, and the diameter of the foundation. The circular foundation will be made up of rectangular foundations. The modelling of a circular foundations is more precise with a large number of foundations. However, calculation time is then longer. The name of the foundation is supplemented with the current number (start number = 1).

The foundation base, the excavation unloading and the foundation stresses will be entered in accordance with the default values in the "**Foundation/Values for all foundations**" menu item. The default values can be adapted via the "**Stresses …**" button. Alternatively, the stresses can be calculated via the vertical load V and the moments M(x) and M(y). Activate the "**Stresses via M and V**" check box. Then enter the corresponding data for the stress analysis via the now active "**M and V**" button.

Immediately after generation, the created foundations can be deleted by pressing the [**Backspace**] key, if the circular foundation generated is not according to your wishes. Optimum circular foundations can be generated with triangular foundations, as adjustment to the foundation geometry is much better using triangles (see "**Triangular foundations/Generate foundations**" menu item, Section 9.5.21).

9.4.11 "Generate annular foundation" menu item

In complete analogy to circular foundations (see Section 9.4.10), you can generate an annular foundation using rectangular foundations.



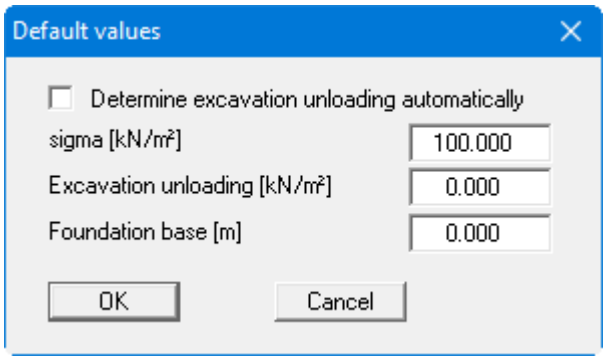
You need only additionally enter the interior diameter. Optimum annular foundations can be generated with triangular foundations, as adjustment to the foundation geometry is much better using triangles (see "**Triangular foundations/Generate foundations**" menu item, Section 9.5.21).

9.5.1 General notes on triangular foundations

The "**Triangular foundations**" menu appears if the appropriate selection has been made in the "**File/New**" menu item (see Section 9.1.1). With this, it is possible to calculate settlements and stresses below a flexible triangular foundation of any shape and linear loading. By combining the triangular foundations in mosaic-like manner, any type of load (with reference to ground plan and load size) can be easily modelled. Further information about triangular foundations can be found in "*Setzungen und Spannungen unter Dreiecksfundamenten*" ("*Settlements and Stresses below Triangular Foundations*"), Dr. Johann Buß, Geotechnik 22 (1999) No. 1.

9.5.2 "Default values" menu item

Here you can specify the foundation stress, excavation unloading and depth of the foundation base. If you prefer, the excavation unloading can be determined automatically. The program automatically calculates the excavation unloading from the depth of the foundation base and the unit weight of the soil.



The unit weight of the soil is defined in the "**Soils/Layers**" menu item (see Section 9.2.1).

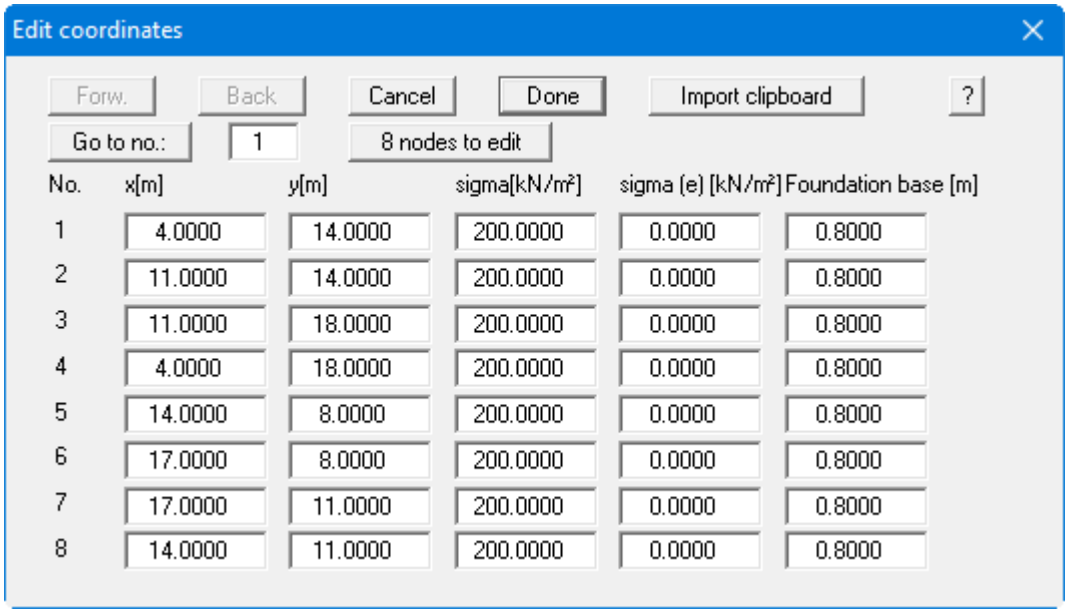
9.5.3 "Define nodes" menu item

Use the left mouse button to define a new foundation node or the right mouse button to delete a previously defined node. Each new foundation node is assigned the default stress (see menu item "**Triangular foundations/Default values**", Section 9.5.2). If a mesh has already been generated using existing foundation nodes, new nodes can only be generated outside of the existing mesh.

In the dialog box for this menu item it is also possible to select whether node numbers, stresses, excavation unloading or foundation base depths are entered at the nodes. A change of visualisation always affects all nodes.

9.5.4 "Edit nodes (via table)" menu item

Using this menu item you can edit the coordinates of the existing foundation nodes or enter new ones. After clicking on it the following dialog box appears:



If the current number of nodes needs to be edited, click the "**x nodes to edit**" button and enter the new number of nodes. It also possible to delete nodes by reducing the number. Any existing foundation mesh is also deleted. Navigate through the table using "**Forw.**" and "**Back**". Each new foundation node is initially assigned the default values for stress, excavation unloading and foundation base (see menu item "**Triangular foundations/Default values**", Section 9.5.2). The data can then be edited in the table.

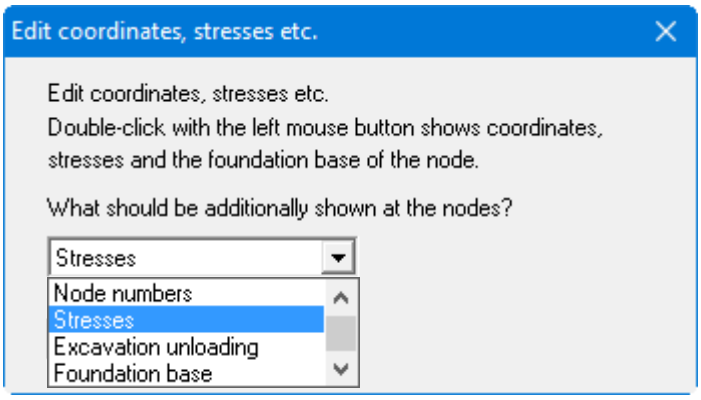
If you already have the coordinates and additional data available in a text editor or in Excel, they can be imported from the Windows clipboard using the corresponding button. Information on this is provided via the "**?**" button.

9.5.5 "Move nodes" menu item

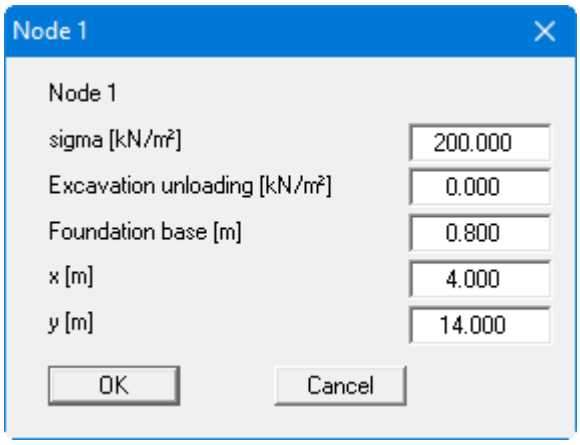
After clicking this menu item an info box is displayed. The foundation nodes can be moved when holding the left mouse button. The coordinates of the current node are displayed in the title bar. The last node movement can be undone using the [**Backspace**] key.

9.5.6 "Edit nodes (individually)" menu item

Using this menu item it is possible to subsequently edit the stress, excavation unloading, depth of the foundation base and the position of individual nodes. First select the information to be entered at the nodes from the dialog box.

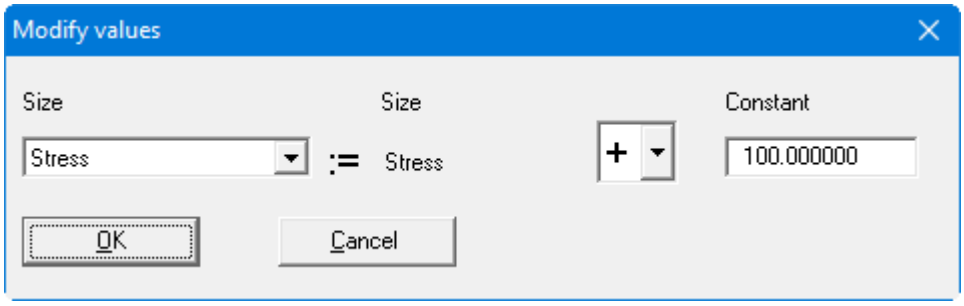


By double-clicking near a node, the following dialog box opens, allowing the coordinates or the stress, excavation unloading and the foundation base depth to be edited.



9.5.7 "Modify values" menu item

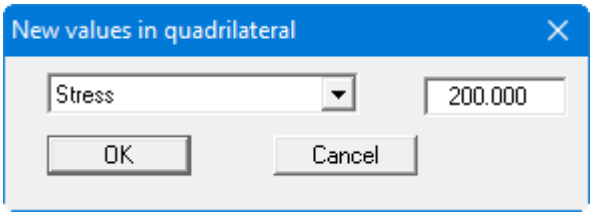
After activating this menu item it is possible to modify subsequently the stresses, excavation unloading or foundation bases at all foundation nodes. The following dialog box appears:



The current values will be modified with a constant in accordance with the selected specified calculation type. The modification is valid for all nodes. This function allows fast editing of the described values with many nodes.

9.5.8 "Assign values in section" menu item

Selected foundation nodes can be subsequently assigned new values for stress, excavation unloading or foundation base. Define an anticlockwise quadrilateral enclosing the nodes to be edited. The following dialog box then opens:



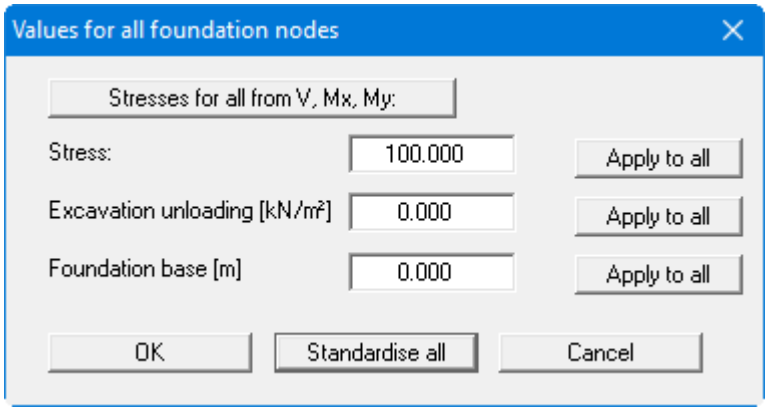
Select the required parameter and enter the new value.

9.5.9 "Generate nodes in array" menu item

Geometric shapes allowing automatic node generation on selected subdivisions can be selected for simplified foundation node input using this menu item. This functions analogous to the menu item "**Mesh/Generate nodes in array**", which is described in Section 9.3.5.

9.5.10 "Values for all foundations" menu item

Using this menu item it is possible to specify stress, excavation unloading and foundations base globally for all foundation nodes. After clicking on it the following dialog box appears:



An "**Apply to all**" button follows each input box. By clicking any button the respective data is adopted for all foundation nodes. You then return to the above dialog box and can edit further data or exit the box via the "**Cancel**" button. The "**Apply to all**" button adopts all data for all foundation nodes. The dialog box is closed automatically after this button is clicked.

The stress can also be determined by pressing the "**Stress for all from V, Mx, My:**" button. A dialog box then opens for entering the data. If the stress is determined from this data the dialog box also closes again automatically.

9.5.11 "Manual mesh" menu item

After input of the foundation nodes this menu item is used to define the mesh. Three nodes must be clicked using the left mouse button. An triangle element can be deleted by selecting the three corresponding nodes once again using the left mouse button.

9.5.12 "Automatic" menu item

After entering the foundation nodes automatic mesh generation can be carried out using this menu item (Delauney triangulation). If a triangle foundation mesh already exists it can be either deleted or supplemented. Only in exceptional cases you should select the "**Supplement**" button, as triangulation follows certain laws which may not allow sensible complementing of a current partial mesh.

9.5.13 "Delete nodes and mesh" menu item

With this menu item you can delete selected foundation triangles. You must first click four points in anti-clockwise direction. All triangles with their centroid within this quadrilateral will be deleted. If the "**Delete nodes and mesh**" button is pressed, the complete mesh and all foundation nodes are deleted.

9.5.14 "Refine individual elements" menu item

Foundation mesh elements can be selected for individual refinement using this menu item, after selecting the refining method.

Three different methods can be used for triangle mesh refinement. Method '**3**' generally gives the best mesh refinement and is recommended. A description of the 3 refinement methods can be found in Section 9.3.11.

9.5.15 "Refine elements in section" menu item

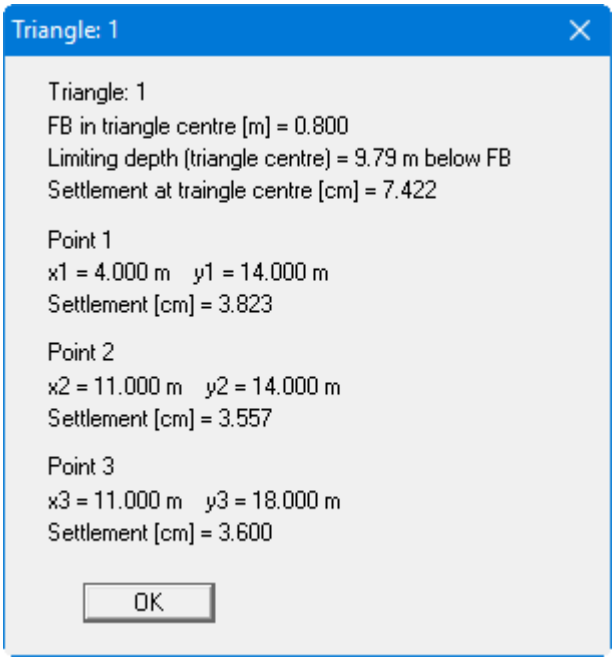
A number of elements of the foundation mesh previously enveloped in a polygon can be refined using this menu item. A description of the 3 refinement methods can be found in Section 9.3.11.

9.5.16 "Refine all elements" menu item

All elements of the triangle foundation mesh can be refined using this menu item. A description of the 3 refinement methods can be found in Section 9.3.11.

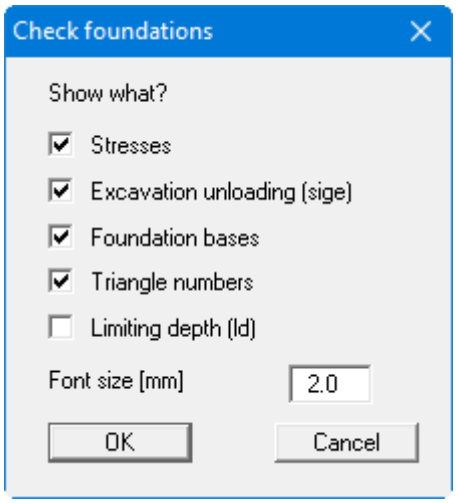
9.5.17 "Results" menu item

The results of the settlement analysis for selected triangular foundations can be viewed in a message box using this menu item. If the system has not yet been analysed, only the triangular foundation point coordinates and foundation base are displayed in the message box. For example, after clicking an analysed triangular foundation the following results are displayed:



9.5.18 "Check foundations" menu item

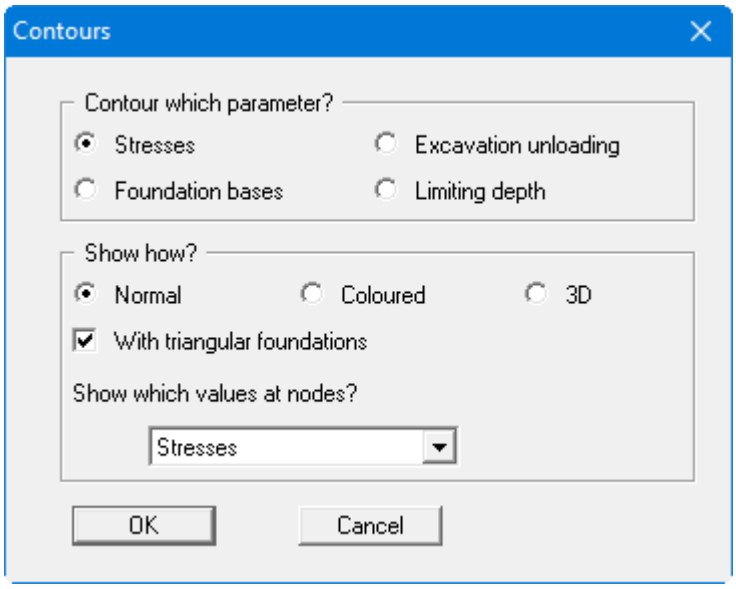
The foundation node input data can be checked via this menu item or, alternatively, by pressing [**F8**]. The individual elements to be visualised can be specified using the check boxes in the dialog box.



Adopt the preferences by pressing "**OK**". The selected data are then displayed in the foundation nodes visualised on the screen. The abbreviations used are described in the ***General legend*** (see Section 9.9.8).

9.5.19 "Contours" menu item

In analogy to the menu item "**Foundations/Stress contours**" the stress distribution in the triangular foundations can be visualised as a coloured contour diagram with the aid of this menu item. The following dialog box opens:



Beside the stresses, the excavation unloading, the foundation bases or the limiting depths can be visualised as contours. Choose between normal line, coloured or 3D visualisation. If the "**With triangular foundations**" check box is activated, the foundation mesh is also visualised. The values given in the option box are entered at the foundation nodes.

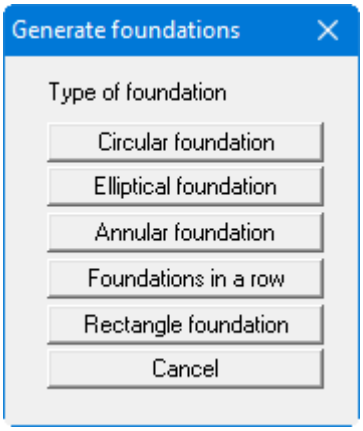
Details of contour visualisation (normal, coloured, 3D) can be found in the menu item "**Evaluation/Settlement contours**" (see Section 9.7.3) in the respective sub-sections.

9.5.20 "Test mesh" menu item

By activating this menu item, the foundation mesh will be investigated with reference to overlapping of individual triangles.

9.5.21 "Generate foundations" menu item

It is possible to automatically generate foundations. First, select the required type of foundation in the following dialog box:



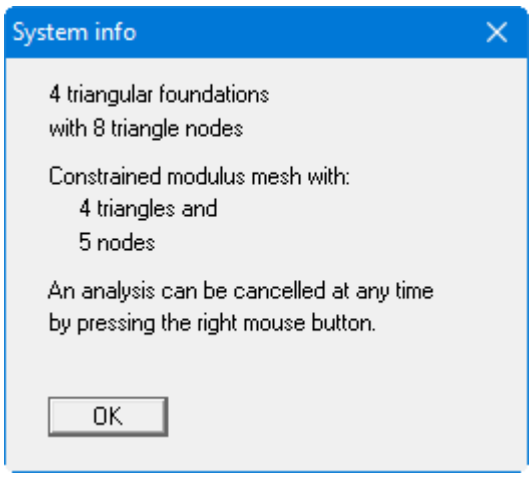
In contrast to the automatic generation of rectangular foundations, the variously shaped foundations are configured using triangular foundations for this menu item. In this way it is possible to model the shape and loading of a foundation much more easily. The "**Rectangular foundation**" function considers, if necessary, the case of a foundation which separates from the soil (foundation gap). If this is the case, a correspondingly smaller foundation will be drawn.

The following foundations can be generated:

- "**Circular foundation**"
  A circular foundation can be compiled from several triangular foundations. Input is made in analogy to the menu item "**Foundations/Generate circular foundation**". A description can be found in Section 9.4.10.
- "**Elliptical foundation**"
  An elliptical foundation can be compiled from several triangular foundations. Input is analogous to the menu item "**Foundations/Generate circular foundation**". Different values must be entered for the diameter in x and y directions. A description can be found in Section 9.4.10.
- "**Annular foundation**"
  An annular foundation can be compiled from several triangular foundations. Input is made in analogy to the menu item "**Foundations/Generate annular foundation**". A description can be found in Section 9.4.11.
- "**Foundations in a row**"
  Several uniformly sized footing foundations can be generated in a row: Input is made in analogy to the menu item "**Foundations/Generate foundations in a row**". A description can be found in Section 9.4.9.
- "**Rectangular foundation**"
  A rectangular foundation can be generated by defining the coordinates of the bottom left corner in the dialog box, as well as the length, breadth and any foundation inclination. In addition, the vertical force and the moments in x and y direction are defined. If base tilt occurs a foundation correspondingly reduced in size will be generated.

9.6.1 "Info" menu item

The following system information e.g., appears:

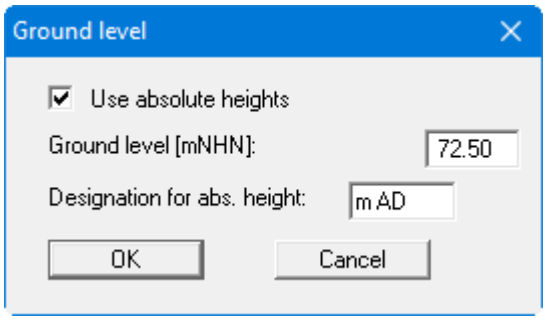


9.6.2 "Project identification" menu item

You can enter a description of the current system; this will then be used in the ***General legend*** (see Section 9.9.8).

9.6.3 "Ground level" menu item

Absolute heights can be adopted using this menu item. Enter the new ground level elevation. You can modify the designation for the absolute elevation (here: m AD) in the following input field.



After leaving the dialog box using "**OK**", the new ground level is shown in the "**Soils/Default layer depths**" dialog box. The previously entered layer bases are automatically converted to absolute depths by the program.

9.6.4 "Limiting depth" menu item

The program can determine the limiting depth using 2 or 3 different procedures, depending on the type of foundation used. The methods were described in detail in the brief introduction (see "**Analysis of rectangular foundations/Specify limiting depth**" in Section 7.2.9 and "**Analysis of triangular foundations/Specify limiting depth**" in Section 7.3.4).

9.6.5 "Analysis options" menu item

Use this menu item to define the type of settlement to be analysed. Three different fundamental options exist, which were described in detail in the brief introduction under "**Analysis of rectangular foundations/Specify type of settlement analysis**" (see Section 7.2.10).

9.6.6 "Analyse" menu item

Analysis of foundation settlements is started via various dialog boxes, depending on the type of foundation selected. The description of analysis of rectangular foundations can be found in "**Analysis of rectangular foundations/Calculate foundation settlements**" (see Section 7.2.11). The description of analysis of triangular foundations can be found in the brief introduction under "**Analysis of triangular foundations/Calculate foundation settlements**" (see Section 7.3.6).

> If you have calculated a system and then edit the foundation data, the program internally deletes all calculated settlements, as they are then no longer valid for the altered system. You should therefore always evaluate such a system first. Alternatively, you have the possibility of saving the calculation data using the "**File/Save**" menu item. Any calculated settlements will be saved with the file and are then available for later evaluation.

9.6.7 "Precision" menu item (for triangular foundations only)

If you are working with triangular foundations, you can specify the precision of the numerical integration using this menu item. A high degree of precision leads to longer calculation times and vice versa. You must weight up between the complexity of the foundation and the desired precision.

9.6.8 "Undo" menu item

If you have carried out any changes to dialog boxes or moved objects to a different position on the screen after selecting the "**Graphics preferences/Move objects**" menu item or using the [**F11**] function key, this menu item will allow you to undo the movements. This function can also be reached by using the key combination [**Alt**] + [**Back**] or the appropriate tool in the toolbar (see Section 9.9.6).

9.6.9 "Restore" menu item

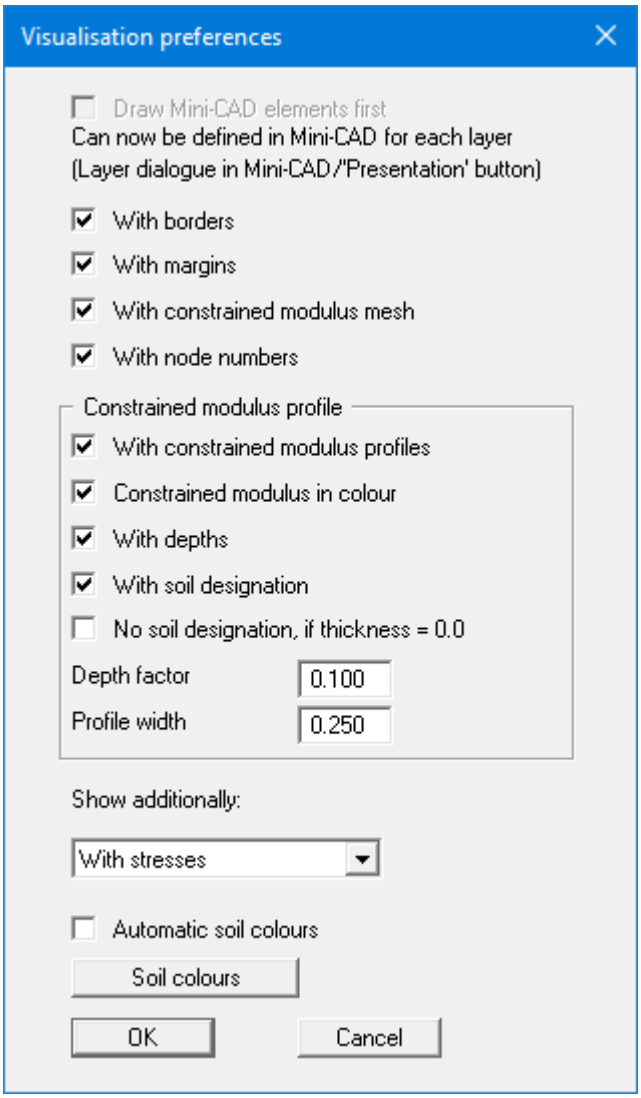
When this menu item is selected the last change made in a dialog box or the last change in the position of objects, which you undid using the menu item "**Page size + margins/Undo**" will be restored. This function can also be reached by using the key combination [**Ctrl**] + [**Back**] or the appropriate tool in the toolbar (see Section 9.9.6).

9.6.10 "Preferences" menu item

You can activate or deactivate the undo functions.

9.7.1 "Preferences" menu item

Graphics visualisation preferences can be specified via this menu item. The following box opens for rectangular foundations:



Objects added using **Mini-CAD** are normally drawn via the system representation. If the **Mini-CAD** objects are to be positioned behind the graphics this can be specified in the pop-up menu of the **Mini-CAD** module (layer dialog in **Mini-CAD**/"**Presentation**" button).

By deactivating the corresponding check boxes in the menu item "**Page size + margins/Page size and margins**" the defined borders and page margins may be switched off (see Section 9.10.5). Visualisation of the constrained modulus mesh and the constrained modulus node numbers can also be deactivated.

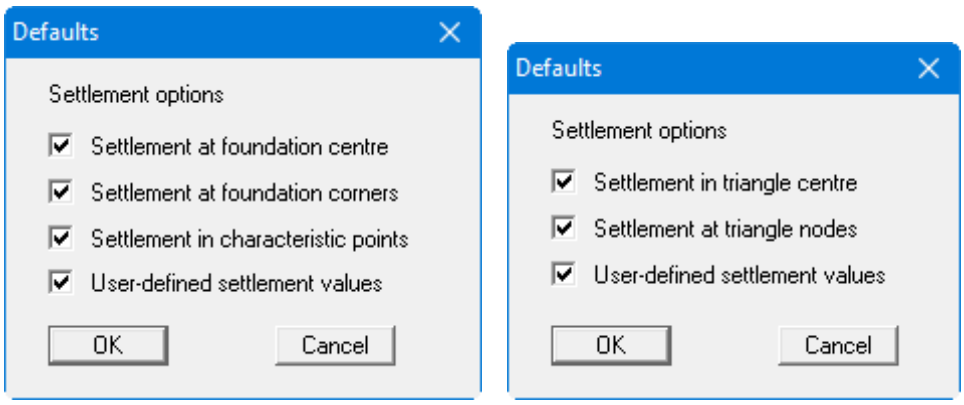
In the "**Constrained modulus profile**" group box, visualisation of the individual profiles can be altered or switched off completely by deactivating the "**With constrained modulus profiles**" check box. "**Depth factor**" allows you to control the length of the profile columns in the presentation.

The foundation name and the stresses on rectangular foundations can also be visualised, as shown above. An option box appears for triangular foundations, instead of the two check boxes, allowing the user to decide whether stress, excavation unloading or the foundation base are displayed at the foundation nodes.

If the "**Automatic soil colours**" check box is activated, the constrained modulus profile layers will be automatically assigned soils colours by the program. If the check box is not selected, the soil colours individually defined using the "**Soil colours**" button will be adopted.

9.7.2 "Settlement location display options" menu item

After going to this menu item, the following dialog boxes open for rectangular and triangular foundations:

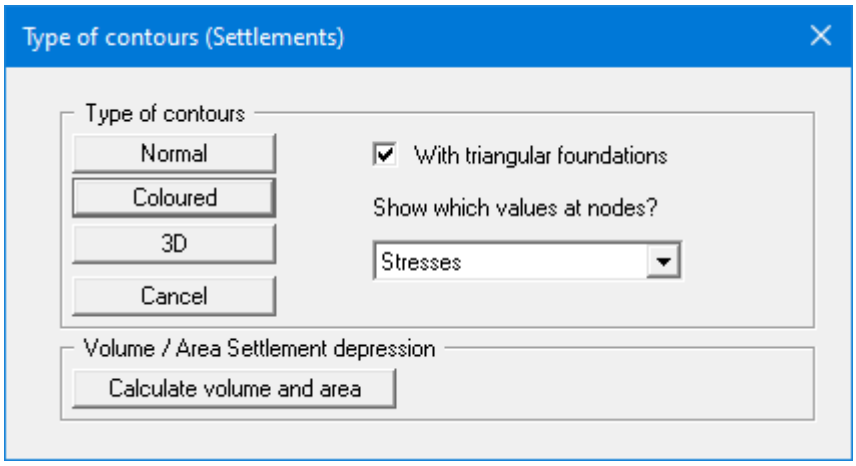


Within this dialog box the presentation parameters for the settlement calculations are specified. If certain settlement values have not yet been calculated, they will naturally not be entered despite being activated.

9.7.3 "Settlement contours" menu item

9.7.3.1 General information on representation of contour lines

This menu item allows visualisation of the analysed settlements in a contour diagram. Further, the program creates a triangle mesh from all analysed settlements. If an analysis has not yet been performed, an error message is displayed. This triangle mesh, which has nothing to do with the interpolation mesh for the constrained modulus profiles, is used by the program as the basis of interpolation for contour calculation. A dialog box opens allowing the type of contours to be selected:

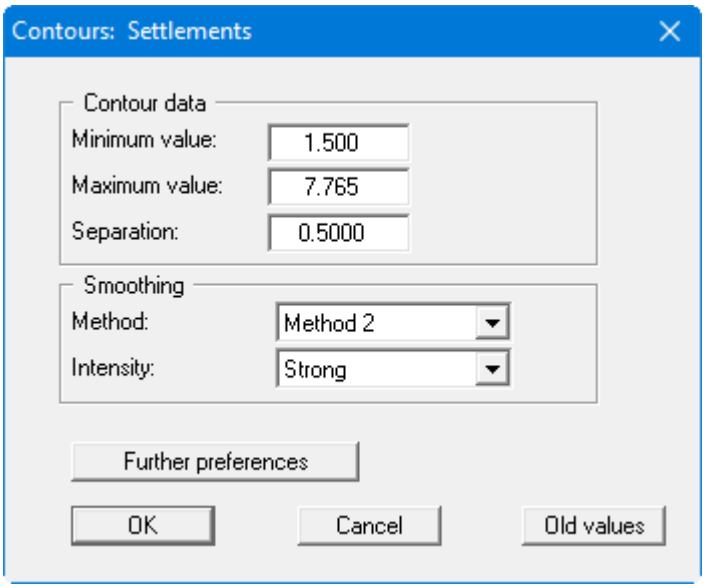


The box above opens for triangular foundations. It is also possible to switch the foundation mesh visualisation on or off. In this case, node labelling can be selected.

By pressing the "**Calculate volume and area**" button a message box opens which contains the area of the triangular mesh from all calculated settlements and the volume of the resulting settlement depression beneath.

9.7.3.2 Normal settlement contours

The following dialog box opens if the "**Normal**" button is pressed:



The program shows the existing smallest and largest settlement values and the separation of the contour lines. If you want the representation to begin with a different value, the initial value can be entered here. You can also vary the spacing, to reduce the number of contour lines drawn for example. The settings displayed here are always those automatically selected by the program. Using the "**Old values**" button, the preferences used for the previous contour line diagram are adopted.

The program supports three smoothing out procedures:

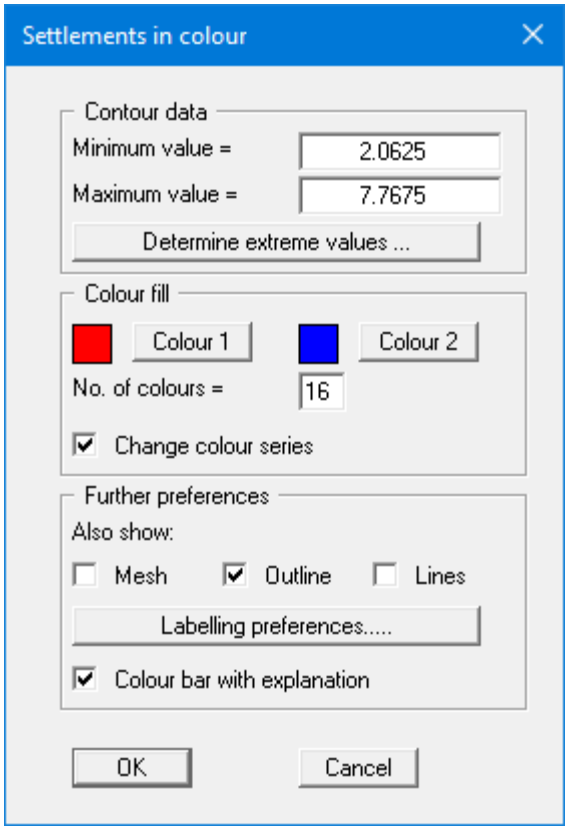
- "**Do not smooth**"
  As a linear method is implemented within the triangle elements, this will provide analysis results without smoothing performed by the program.
- "**Method 1**" and "**Method 2**"
  these smoothing out procedures employ two different Bezier splines. Smoothing out intensities can be given for to acquire smooth contour lines. **Method 2** creates very "**round**" contours, with the danger that the true results may be falsified.

You can specify the type of visualisation for the contour lines by means of the "**Further preferences**" button, e.g. system boundaries, triangle mesh, contour labelling, font size, etc.

If your preferences are confirmed by pressing "**OK**" the contours are displayed on the screen. The colour and pen width defined in "**Graphics preferences/Pen colour and width**" is used. If the contours are to be visualised in a different colour or pen width, they can be edited in this menu item (see Section 9.9.3).

9.7.3.3 Coloured settlement contours

In analogy to the previous section, colour-filled contours can also be created. The following dialog box appears:



- "**Contour data**" group box
  The "**Determine extreme values**" button computes the minimum and maximum settlements and enters them into the appropriate input boxes. However, you need not retain these values, but can also define your own. No smoothing of the contour lines can be performed here.

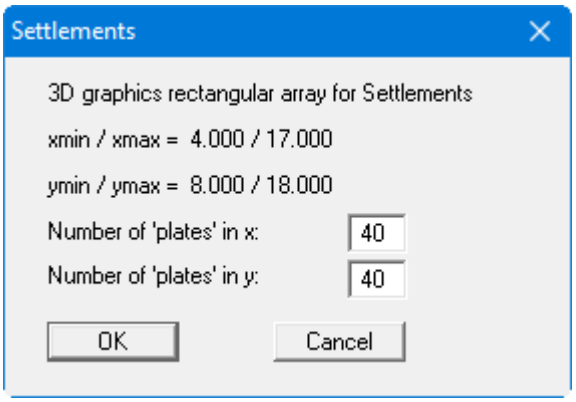
- "**Colour fill**" group box
  You can control the colour subdivisions of the contour diagram using "**No. of colours**". In the example above, 16 colours will be displayed between "**Colour 1**" and "**Colour 2**". The default setting is a colour course from red to blue. These colours can be edited as required after selecting the "**Colour 1**" and "**Colour 2**" buttons, or simply reverse the choice by selecting the "**Change colour series**" check box.

- "**Further preferences**" group box
  In addition to the colour presentation you can also have the triangle mesh and/or the outline displayed. Additional contour lines can also be drawn. Line labelling preferences can be defined by means of the "**Labelling preferences**" button. Here you can also define the font size for the colour bar on the right-hand edge of the screen, which is used to assign the respective colour to the corresponding size on your output sheet. Activate the check box for labelling the colour bar with the explanation of the value displayed.

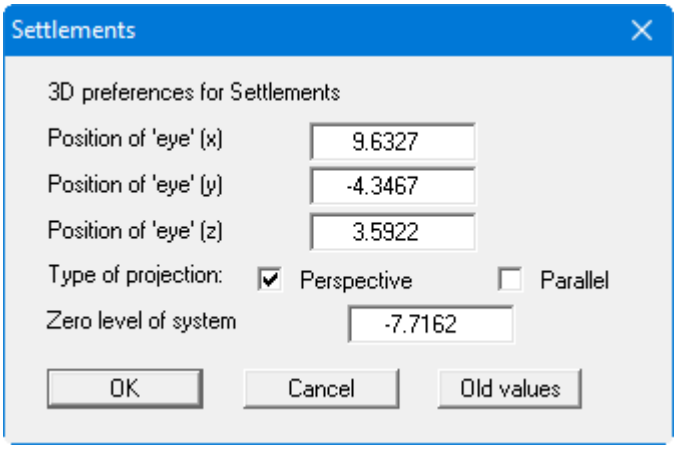
- "**OK**"
  The colours will be drawn after confirmation. If this colour bar is drawn in the right page margin, specify a larger value for the right plotting margin (e.g. 25 mm) in the "**Page size + margins/Page size and margins**" menu item (see Section 9.10.5).

9.7.4 3D settlement contours

This menu item allows presenting contours as a three-dimensional image in a rectangular array. In the following dialog box, you can define the array by selecting the number of x and y subdivisions (plates):

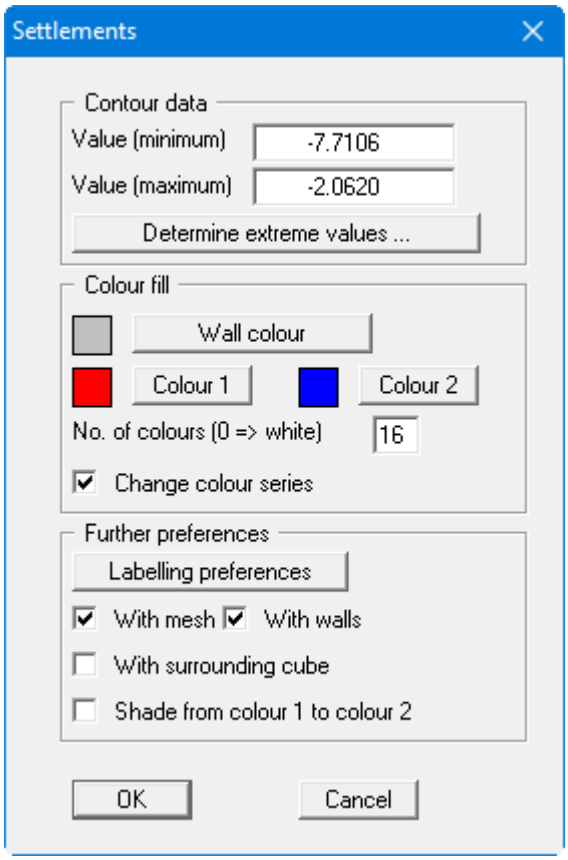


The program interpolates the settlement values to be represented at the array points from the underlying triangle mesh. After leaving the dialog box by pressing "**OK**" you will see the following box:



Here, select the point of observation (position of the ***eye*** in x, y and z coordinates), the type of projection and the zero level of the 3D presentation. The settlements are displayed above and below the zero level (z ordinate) in three dimensions. The program will generally make sensible suggestions for the input values in the dialog box, so you will not normally need to make alterations.

After leaving the dialog box by pressing "**OK**" you will see the following dialog box. It greatly resembles the box shown in the "**Evaluation/Coloured contours**" menu item (see Section 9.7.3.3).



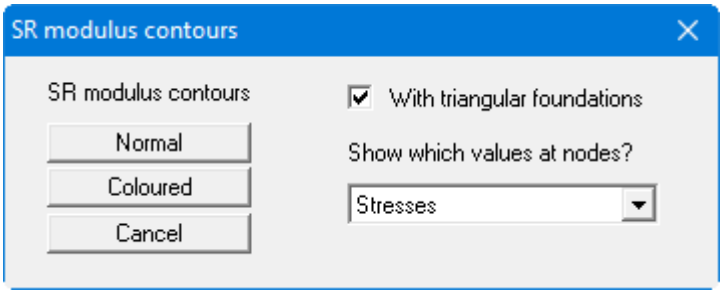
You can also specify whether an enclosing cube should be displayed; in some cases this improves the 3D effect. The "**Shade from colour 1 to colour 2**" check box represents a bit of a special effect. If this check box is activated a light source is simulated in the region of the eye. The angle between the light beam and the respective 3D surface represents a measure of the reflection. The area is shaded in accordance with colours 1 and 2. A good choice of colours, for example, would be dark grey for colour 1 and pale grey for colour 2.

After confirming by pressing "**OK**", 3D graphics are displayed. Parallel to this, a pop-up menu opens at the bottom left; its functions allow the graphics to be rotated (see menu item "**Graphics preferences/3D toolbar**", Section 9.9.7).

9.7.5 "Subgrade reaction contours" menu item

This menu item allows the presentation of the subgrade reaction moduli from calculated settlements in a contour plan. Further, the program creates a triangle mesh from all calculated settlements. However, only those settlement points which lie within the foundation are considered, as the subgrade reaction modulus results from the division of the foundation stress at the point in question by the corresponding settlement value. The thus created triangle mesh, which has nothing to do with the interpolation mesh for the constrained modulus profiles, is used by the program as the basis of interpolation for contour calculation.

After going to this menu item a dialog box opens which is analogous to the settlement contour visualisation dialog box.

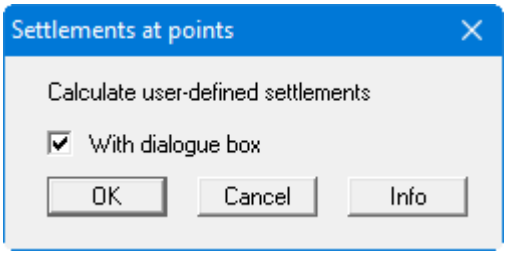


Visualisation of the foundation mesh and node labelling can be selected for triangular foundations. Otherwise, you have two buttons available for the type of contour visualisation:

- "**Normal**"
  The preferences for normal contour visualisation are analogous to the descriptions in Section 9.7.3.2.
- "**Coloured**"
  The preferences for coloured contour visualisation are analogous to the descriptions in Section 9.7.3.3.

9.7.6 "Settlements at points" menu item

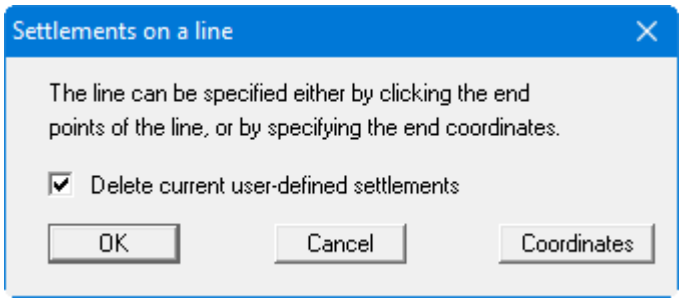
This menu item allows calculation of user-defined settlements at any point within the triangle mesh. Click on the point with the left mouse button. To assist in orientation the current coordinates are shown in the status bar.



If the "**With dialog box**" check box is activated, the coordinates and the analysed settlements are displayed in a message box immediately after clicking the point. This message box must then be closed by pressing "**OK**" to return to the screen graphics and determine the settlements at further points.

9.7.7 "Settlements on a line" menu item

This menu item supports analysis of user-defined settlements along a user-defined line. The line is determined with the left mouse button. For this, two points must be clicked. The user must then enter the number of subdivisions. The settlements along the line will then be calculated in accordance with this number.



If you exit the box by pressing "**OK**", the line is defined by clicking two points with the left mouse button and then entering the number of subdivisions. The settlements along the line are calculated at these points. However, it is also possible to define the two line end points by specifying their coordinates. To do this, click the "**Coordinates**" button. If user-defined settlements have already been calculated, the above check box is also displayed. If you would like to use the previously calculated settlements, deactivate the check box.

9.7.8 "Settlements in quadrilateral array" menu item

This menu item is comparable to the previous one. You must simply define a quadrilateral array with the mouse. You will then be asked to enter a array in x and y direction, at the intersections of which settlement analyses will be carried out.

9.7.9 "Delete user-defined points" menu item

By selecting this menu item you can delete all current user-defined settlement points, after a safety request.

9.7.10 "Save settlement points" menu item

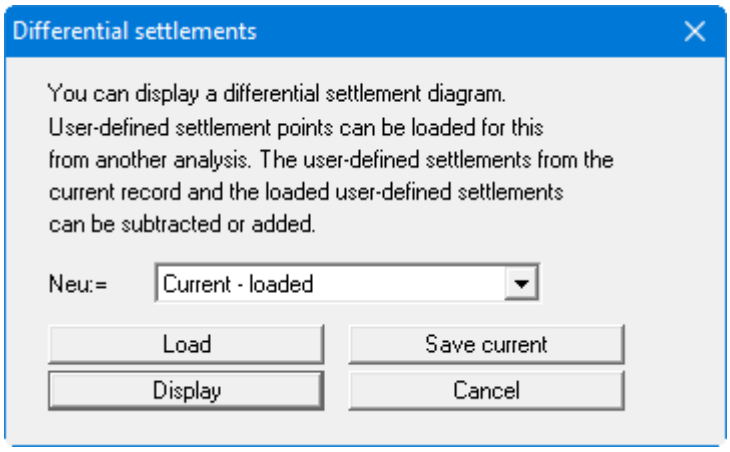
By selecting this menu item, you can save all current user-defined settlement points to a file. A file requester box appears, in which you can enter the file name. The file suffix should be "**.pkt**".

9.7.11 "Load settlement points" menu item

This menu item allows the point files saved using the menu item "**Evaluation/Save settlement points**" to be opened. To reanalyse the settlements at the imported points, go to "**System/Analyse**" and activate the "**Calculate settlements at user-defined points**" check box in the dialog box.

9.7.12 "Differential settlements" menu item

If settlements have been analysed in the current file, this menu item can be utilised to load the settlements from a previous analysis and to generate and visualise the difference or sum total from the old and the new data. The coordinates of the settlement points must be the same in both files. It is therefore useful to save the points of a first settlement analysis using the menu item "**Evaluation/Save settlement points**" to a file which is then used as the basis for a further settlement analysis. The following dialog box opens:



Select the old "**.pkt**" file using the "**Load**" button. The current data can be saved again using the "**Save current**" button, if this has not already been done via "**Evaluation/Save settlement points**". Select the type of difference or sum formation in the option box. The following options are available:

- "**Current - loaded**"
- "**Loaded - current**"
- "**Current + loaded**"

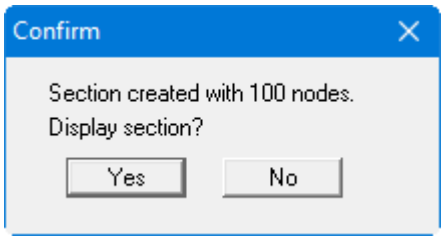
If a file has been loaded the "**Display**" button is active and the previously seen dialog box for contour type selection opens. Normal, coloured or 3D contour visualisation of differential settlements can then be generated (see description of contour visualisation in Sections 9.7.3.2 to □).

9.7.13 "Define multi-node section" menu item

It is possible to visualise the computed, user-defined settlements graphically as a settlement depression. A user-defined section can be specified using this menu item. Left-click the required settlement points, displayed on the screen. If settlement values have not yet been calculated it is, of course, not possible to define a section. After defining the section, visualisation of the settlement depression can be achieved using the menu item "**Evaluation/Display settlements in section**" (see Section 9.7.16).

9.7.14 "Automatic multi-node section" menu item

This menu item allows automatic specification of a section course. If you agree to the initial query, all calculated point settlements are correlated in the specified sequence to form a section. A message on the number of nodes used is then displayed:



If "**Yes**" is clicked, the settlements in section are displayed.

9.7.15 "Display multi-node section course" menu item

The location of the section in plan can be displayed again using this menu item. First, a dialog box opens allowing the settlements to be selected for visualisation. The colour and width of the pen for the section in the screen graphics can be defined in the menu item "**Graphics preferences/Pen colour and width**" (see Section 9.9.3).

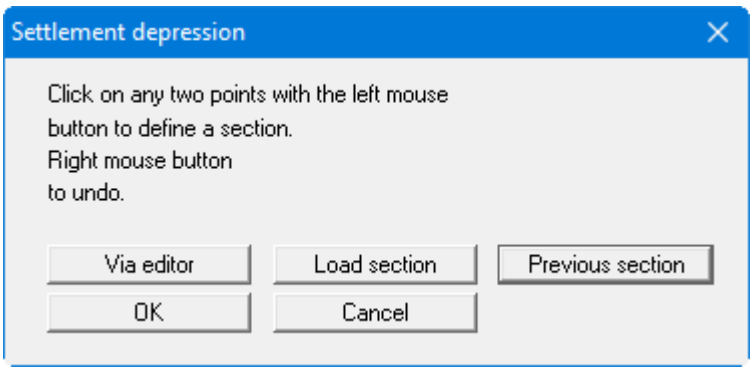
The same visualisation is duplicated in a smaller size in the ***Section course legend***, which is automatically displayed during section visualisation (see Section 9.9.10).

9.7.16 "Display settlements in section" menu item

The settlements within the defined section (using the menu item "**Evaluation/Define multi-node section**" and "**Evaluation/Automatic multi-node section**") are visualised as a settlement depression after going to this menu item. The program automatically optimises the coordinate system used.

9.8.1 "Settlement depression section" menu item

This menu item allows the calculation and presentation of settlement depressions. In contrast to the settlement depression, defined using the menu item "**Evaluation/Define multi-node section**" by connecting previously analysed settlement points, this vertical section is always along a straight line. This is, on the one hand, a restriction but, on the other hand, offers more clarity, as the soil stratification and positions of foundations are shown in such a section. In principle, this is the menu item of choice when it comes to displaying settlement depressions. Specify the means of defining the section in the dialog box.



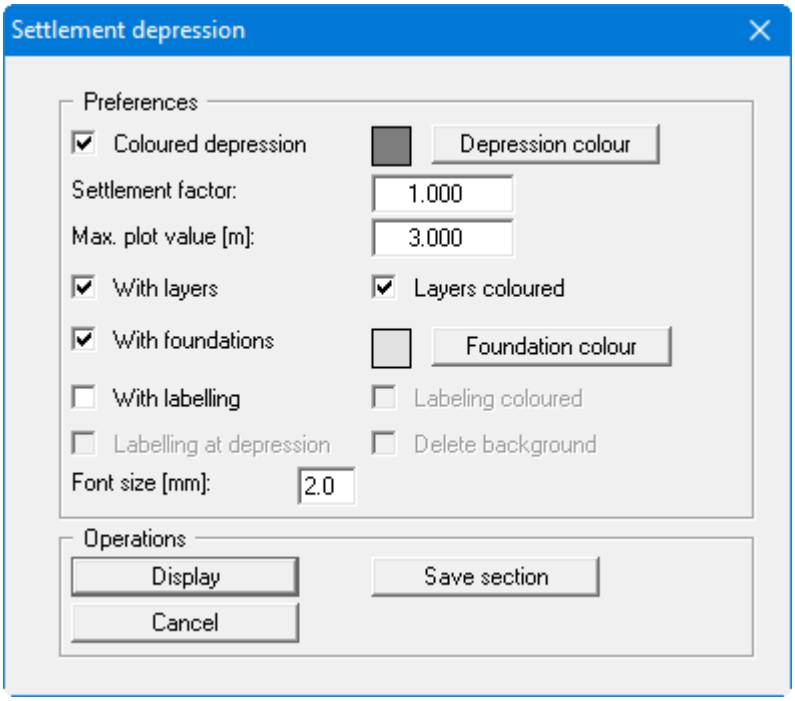
For this type of evaluation the section course is no longer bound to the settlement points, but consists of a start and an end point. Using the "**Via editor**" button you can specify the start and end points using numerical values instead of the mouse. If you have previously saved a section it can be loaded again using "**Load section**". The "**Previous section**" button is only visible if a section has already been defined and loads the section defined immediately prior.

If you click "**OK**" you define a new section by clicking the start and end points using the mouse. Once the end point is defined a dialog box opens for input of the number of subdivisions. The number of subdivisions defines the number of drawdown points for calculation.

Once the section has been defined a dialog box opens allowing preferences for the graphical visualisation of the settlement depression to be specified. The same dialog box can also be accessed via the menu item "**Special/Depression section preferences**". A description can be found in Section 9.8.2. The settlement depression is visualised when you exit the preferences dialog box via the "**Display**" button.

9.8.2 "Depression section preferences" menu item

Preferences for settlement depression visualisation can be specified using this menu item. The following dialog box opens:



With "**Depression colour**" you define the colour with which the depression will be filled when the "**Coloured depression**" check box is activated. The settlement depression will be displayed in a vertical section, generally with the strata (constrained modulus profiles). With "**Settlement factor**" you can influence the depth of the settlement depression in the presentation. With "**Max. plot value**" you specify the presentation of the foundation stress in section. With "**Foundation colour**" you define the colour with which the foundations will be filled when the "**With foundations**" check box is activated.

Activating the "**With labelling**" check box automatic section labelling can be used. When activated the program inserts a table below the section, containing the calculated settlements at the settlement points defined. The labelling font size can be altered to suit requirements. If the "**Labelling coloured**" check box is activated the settlements are visualised in the above defined colour for the depression. If the "**Labelling at depression**" check box is activated, the automatic labelling table is no longer shown and the settlements entered in the graphics directly at the settlement points.

If the specified section is required again later for further evaluations, you can save this section to a file and reload it again later using the above shown dialog box. After leaving the dialog box via the "**Display**" button the program first calculates the settlements and then displays the settlement depression for the specified points.

A location sketch is automatically displayed on the screen showing the system outline and the defined section. This section course legend is activated by default, but can be deactivated in the "**Graphics preferences/Section course legend**" menu item (see Section 9.9.10).

9.8.3 "Display section course (settlement depression)" menu item

The location of the section in plan can be displayed again using this menu item and be printed as an appendix to the report, for example. The same visualisation is duplicated in a smaller size in the ***Section course legend***, which is automatically displayed during section visualisation (see Section 9.9.10).

9.8.4 "Stress section" menu item

This menu item allows the analysis and visualisation of stress distributions in vertical sections. The procedure is almost completely analogous to the procedure for defining and subsequently visualising settlement depressions. A dialog box opens for defining the sections, as shown in Section 9.8.1.

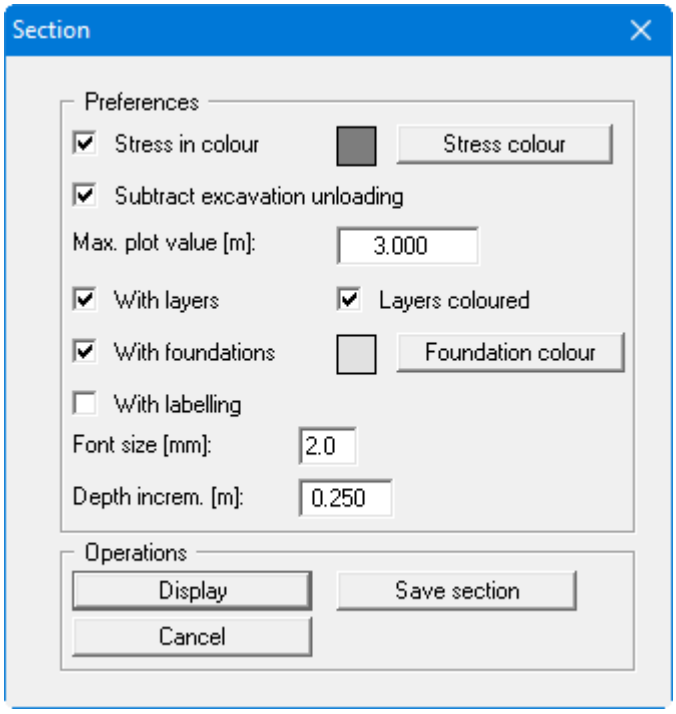
For this type of evaluation the section course is no longer bound to the settlement points, but consists of a start and an end point. Using the "**Via editor**" button you can specify the start and end points using numerical values instead of the mouse. If you have previously saved a section it can be loaded again using "**Load section**". The "**Previous section**" button is only visible if a section has already been defined and loads the section defined immediately prior.

If you click "**OK**" you define a new section by clicking the start and end points using the mouse. Once the end point is defined a dialog box opens for input of the number of subdivisions. The number of subdivisions defines the number of drawdown points for calculation.

Once the section has been defined a dialog box opens allowing preferences for the graphical visualisation of the stress section to be specified. The same dialog box can also be accessed via the menu item "**Special/Stress section preferences**". A description can be found in Section 9.8.5. The stress section is visualised when you exit the preferences dialog box via the "**Display**" button.

9.8.5 "Stress section preferences" menu item

Preferences for stress section visualisation can be specified using this menu item. The following dialog box opens:



With "**Stress colour**" you define the colour with which it will be filled when the "**Stress in colour**" check box is activated. "**Subtract excavation unloading**" allows you to select calculations with or without excavation unloading. With "**Max. plot value**" you specify the presentation of the foundation stress in section. You can specify whether the soil stratification is to be shown. If the "**Layers coloured**" check box is activated, the soil layers will be colour-filled in accordance with the specified soil colours (see menu item "**Evaluation/Preferences**", Section 9.7.1). With "**Foundation colour**" you define the colour with which it will be filled when the "**With foundations**" check box is activated. Using the "**Labelling**" check box you can have the stress distribution labelled. With the "**Depth increment**" value you define the vertical spacing for stress calculations. Very small value mean especially long calculation times. Very large values can lead to an ***unround*** stress course.

If the specified section is required again later for further evaluations, you can save this section to a file and reload it again later using the above shown dialog box. After leaving the dialog box via the "**Display**" button the program displays the stress distribution.

A location sketch is automatically displayed on the screen showing the system outline and the defined section. This section course legend is activated by default, but can be deactivated in the "**Graphics preferences/Section course legend**" menu item (see Section 9.9.10).

9.8.6 "Display section course (stress section)" menu item

The location of the stress section in plan can be displayed again using this menu item and be printed as an appendix to the report, for example. The same visualisation is duplicated in a smaller size in the ***Section course legend***, which is automatically displayed during section visualisation (see Section 9.9.10).

9.8.7 "Stress bulb section" menu item"

The stresses for any given vertical section can be visualised as contours using this menu item. The same dialog boxes open as shown in "**Special/Stress section**" in Section 9.8.4. However, after leaving the graphical preferences dialog box by using the "**Display**" button you select the type of contours.

Once you have selected the type of contours required the program calculates the stresses along the specified section. You then see the familiar dialog boxes for the normal or the coloured contour line diagram (see Sections 9.7.3.2 and 9.7.3.3).

9.8.8 "Normal contours (stress bulb)" menu item

If the stresses for the specified section have already been calculated using the previous menu item "**Special/Stress bulb section**" you can move directly to the normal contour diagram using this menu item. The familiar dialog box opens (see Section 9.7.3.2).

9.8.9 "Coloured contours (stress bulb)" menu item

If the stresses for the specified section have already been calculated using the menu item "**Special/Stress bulb section**" you can move directly to the coloured contour diagram using this menu item. The familiar dialog box opens (see Section 9.7.3.3).

9.8.10 "Display section course (stress bulb)" menu item

The location of the stress bulb section in plan can be displayed again using this menu item and be printed as an appendix to the report, for example. The same visualisation is duplicated in a smaller size in the ***Section course legend***, which is automatically displayed during section visualisation (see Section 9.9.10).

9.8.11 "Vertical settlements section" menu item

Specification of the section for displaying the vertical settlements is in analogy to the menu item "**Special/Stress section**" (see Section 9.8.4). If you close the dialog box for the graphical representation preferences using the "**Display**" button, the program first calculates the settlements in the user-defined section. You will then see a combo box for selecting the type of contour representation.

Once you have selected the type of contours required the program calculates the settlements along the specified section. You then see the familiar dialog boxes for the normal or the coloured contour line diagram (see Sections 9.7.3.2 and 9.7.3.3).

9.8.12 "Normal contours (vertical settlements)" menu item

If the settlements for the specified section have already been calculated using the previous menu item "**Special/ Vertical settlements section**" you can move directly to the normal contour diagram using this menu item. The familiar dialog box opens (see Section 9.7.3.2).

9.8.13 "Coloured contours (vertical settlements)" menu item

If the settlements for the specified section have already been calculated using the menu item "**Special/ Vertical settlements section**" you can move directly to the coloured contour diagram using this menu item. The familiar dialog box opens (see Section 9.7.3.3).

9.8.14 "Display section course (vertical settlements)" menu item

The location of the vertical settlements in plan can be displayed again using this menu item and be printed as an appendix to the report, for example. The same visualisation is duplicated in a smaller size in the ***Section course legend***, which is automatically displayed during section visualisation (see Section 9.9.10).

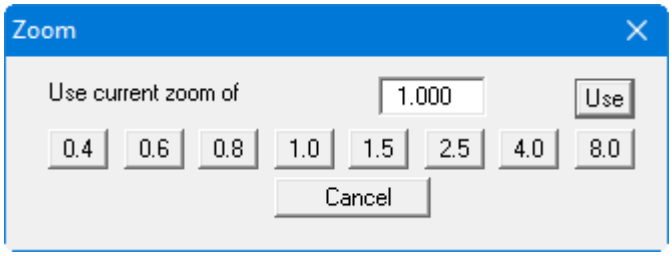
9.8.15 "Stresses for GGU-CONSOLIDATE" menu item

If a stress section was previously defined (see Section 9.8.4), the stress distributions at the analysed points can be exported to the **GGU-CONSOLIDATE** program. Select the required point in a dialog box and save the stress distribution to a "**.kon_spg**" file.

9.9.1 "Refresh and zoom" menu item

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

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



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

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

9.9.2 "Zoom info" menu item

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

9.9.3 "Pen colour and width" menu item

For reasons of clarity different colours are used as default for the graphical representation of drawdowns, contour lines, groundwater, etc. You can edit the pen widths for the elements shown in the dialog box; by clicking on the button with the element designation you can also edit the pen or fill colours

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

9.9.4 "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.

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

Using these two menu items 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. The same pop-up menu opens for both menu items, the icons and functions used are described in more detail in the **Mini-CAD** manual saved in the '**C:\Program Files (x86)\ GGU-Software\ Manuals**' folder during installation. The differences between the Mini-CAD and Header CAD are as follows:

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

9.9.6 "Toolbar preferences" menu item

After starting the program a horizontal toolbar of 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-SETTLE.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) of 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 can 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 6.5).

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

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

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

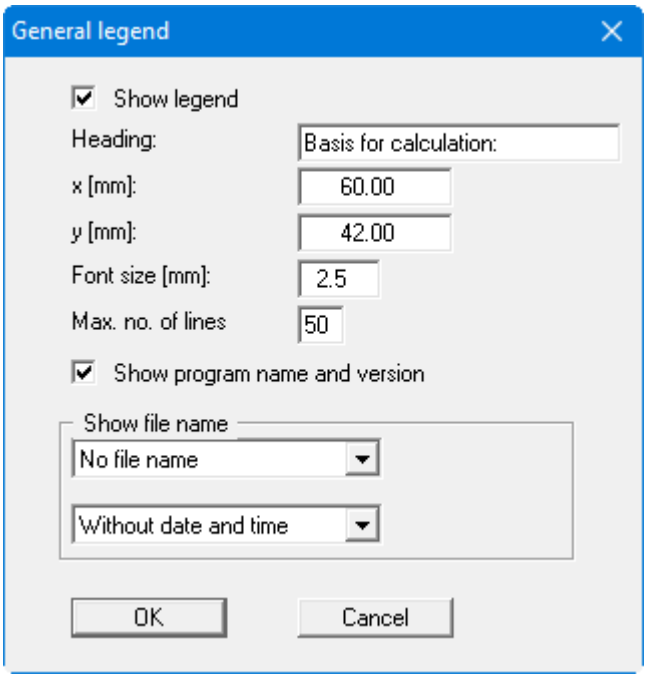
9.9.7 "3D toolbar" menu item

This menu item allows you to choose a pop-up window with tools for rotating and zooming three dimensional images.

The tools in this pop-up 3D window allow you to rotate the graphics around one of the three axes. The plus and minus signs designate the direction of rotation. The angle of rotation (default: 45°) can be adjusted as wished using the  tool.

9.9.8 "General legend" menu item

A legend with general properties will be displayed on your output sheet if you have activated the "**Show legend**" check box. Using this menu item you can alter the type of presentation. The following dialog box can also be accessed by double-clicking in the legend box.



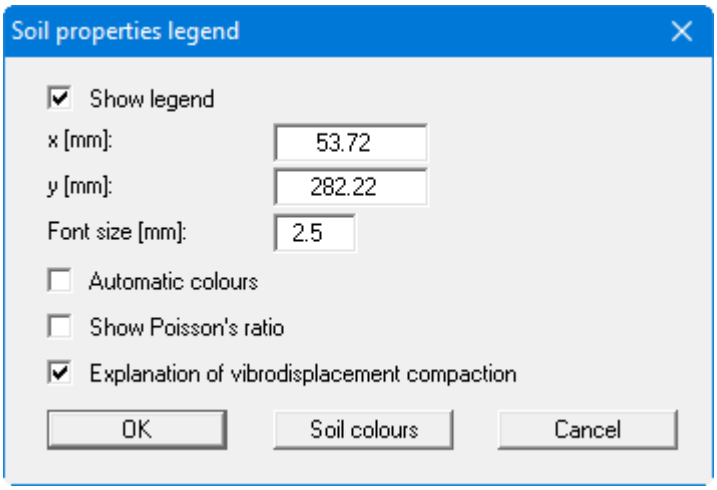
Besides the heading this legend contains information on the current graphics and preferences. You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns 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.

In the ***General legend*** you can, if wished, display information on the program (name and version) and on the current file (name, path, date and time info). Any project identification entered (see Section 9.6.2) will also be shown in the general legend.

After going to the menu item "**Foundations/Check foundations**" or "**Triangular foundations/Check foundations**" the abbreviations used in the graphics are also explained in the legend (see Section 9.4.5 or Section 9.5.18).

9.9.9 "Soil properties legend" menu item

A soil properties legend is shown on the screen when you activate the "**Show legend**" switch. The format and appearance of the legend can be altered using this menu item. The following dialog box can also be accessed by double-clicking in the legend box.



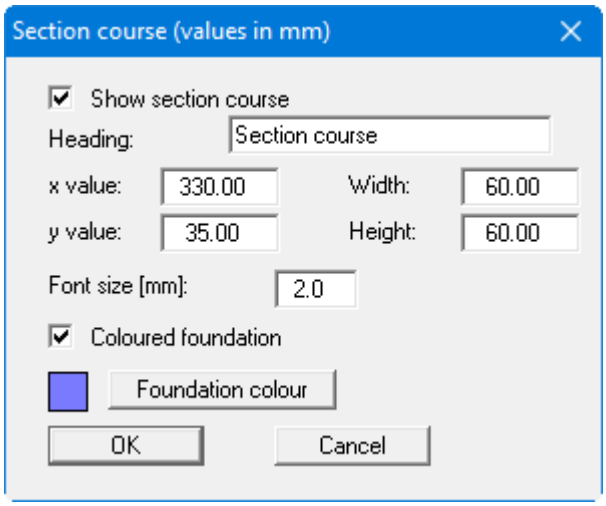
You can define and edit the position of the legend using the values "**x**" and "**y**". The size of the legend is controlled by the values for "**Font size**". The fastest way to modify the position of the legend is to press the [**F11**] function key and then to pull the legend to the new position with the left mouse button pressed.

If you deactivate the "**Automatic colours**" check box, the soil colours will be selected according to your individual preferences, which you can edit using the "**Soil colours**" button.

Poisson's ratio can be switched off in the legend, for example if Poisson's ratio nu = 0.0 for all soils (see Section 9.2.1). It is also possible to display an explanation of the vibrodisplacement compaction in the legend by activating the corresponding check box

9.9.10 "Section course legend" menu item

If a section is visualised a legend is shown which includes a section location sketch. If the "**Show section course**" check box is activated the presentation format can be edited in the dialog box for this menu item. The following dialog box can also be accessed by double-clicking in the legend box.



The position of the legend can be defined and edited using the values "**x**" and "**y**", "**Width**" and "**Height**". You can edit the font size of the entered heading. If the foundations are to be shown in the drawing, activate the corresponding check box and specify the required colour via the "**Foundation colour**" button.

In contrast to the section visualisation in the screen graphics, for example using the menu item "**Special/Display section course**" (see Section 9.8.3), the x and y ordinates of the end points of the section are also displayed as numerical values in the ***Section course legend***.

9.9.11 "Move objects" menu item

When you go to this item you can move the various objects 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.

9.9.12 "Load graphics preferences" menu item

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

9.9.13 "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-SETTLE.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.

9.10.1 "Auto-resize" menu item

This menu item provides a to-scale visualisation, in both x and y coordinates, of the system and result graphics. If you have previously altered the image coordinates using "**Page size + margins/Manual resize (mouse)**" or "**Page size + margins/Manual resize (editor)**", you can quickly achieve a complete view using this menu item. This function can also be accessed using the [**F9**] function key.

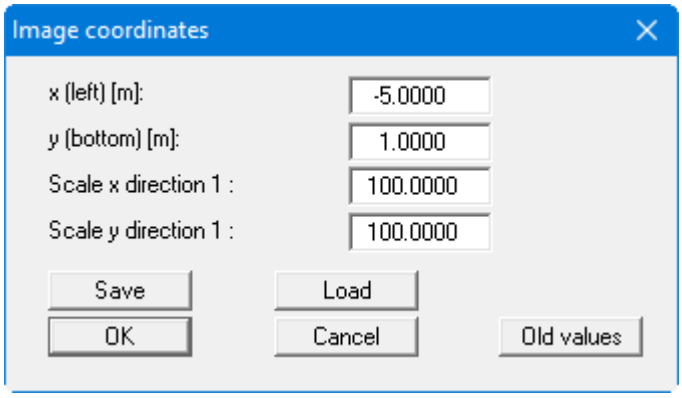
9.10.2 "Manual resize (mouse)" menu item

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

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

9.10.3 "Manual resize (editor)" menu item

You can alter the image coordinates by direct numerical input in a dialog box. This allows precise scale input. The coordinates refer to the ***drawing area***. This can be defined in the "**Page size + margins/Page size and margins**" menu item by means of the plot margins (see Section 9.10.5).



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

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

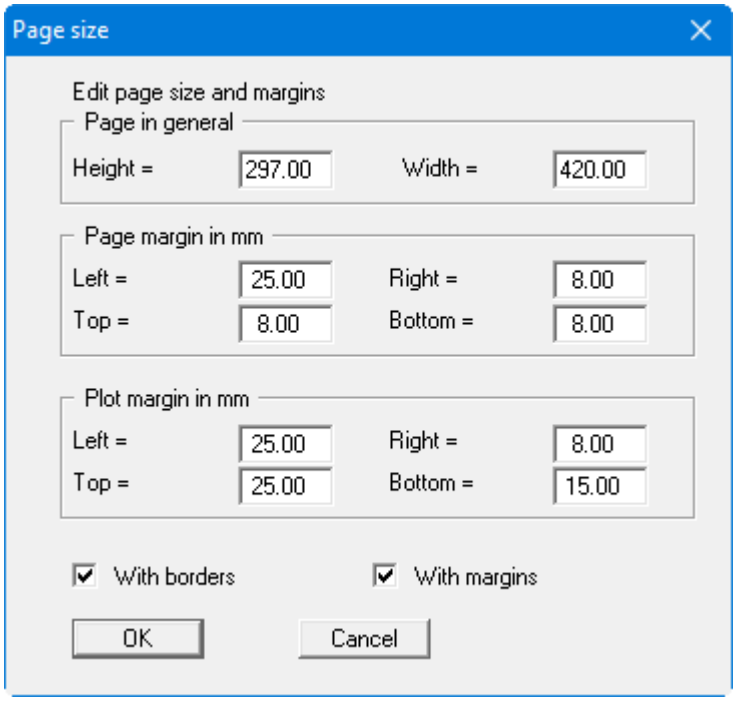
9.10.4 "Font size selection" menu item

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

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

9.10.5 "Page size and margins" menu item

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



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

9.11.1 "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-SETTLE**.

9.11.2 "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. On the "Changelogs" subpage, you can also subscribe to an e-mail notification that informs you of all changes monthly.

9.11.3 "GGU support" menu item

This menu takes you directly to the GGU-Software Support Portal.

9.11.4 "Maxima" menu item

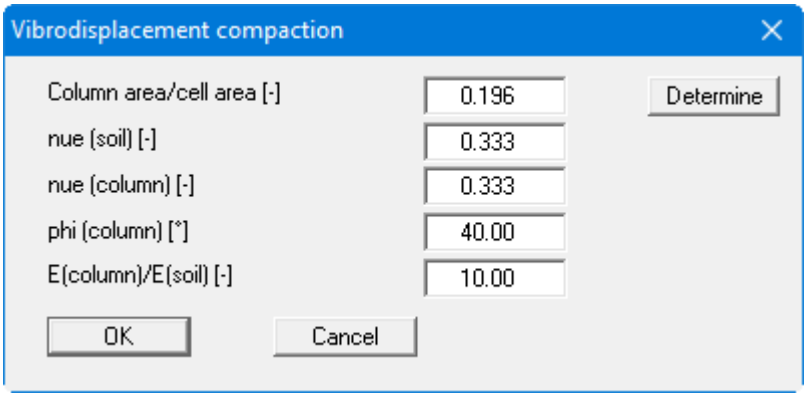
The program default maxima are shown.

9.11.5 "Help" menu item

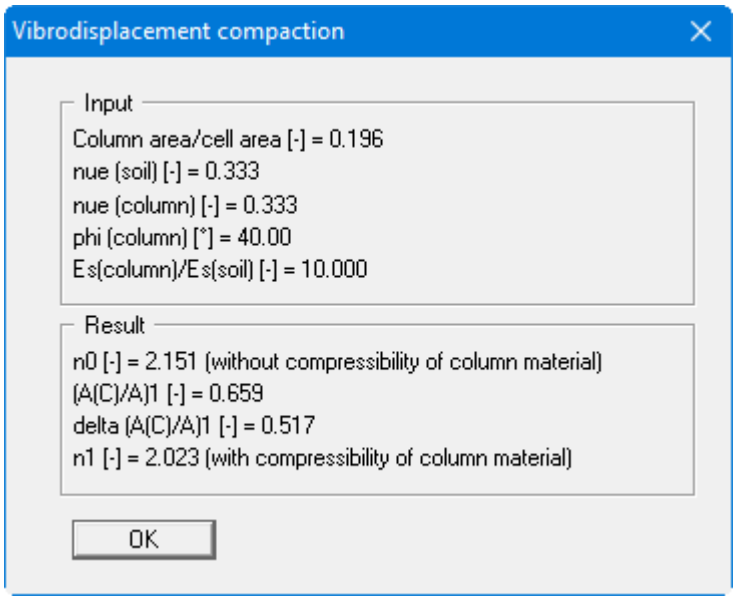
The **GGU-SETTLE** manual is opened as a PDF document. The help function can also be accessed using the [**F1**] function key.

9.11.6 "Test vibrodisplacement compaction" menu item

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



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 presented in a message box:



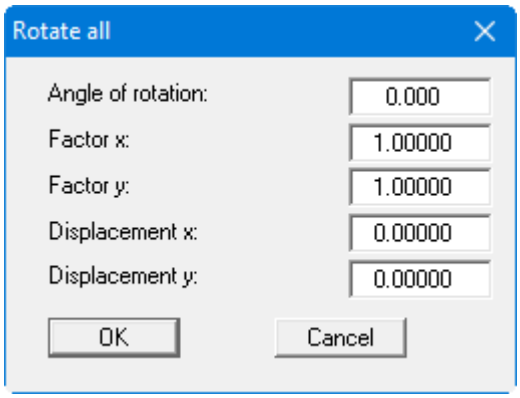
Click "**Cancel**" in the above dialog box to end the menu item.

9.11.7 "What's new?" menu item

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

9.11.8 "Transform all" menu item

This menu item allows the simultaneous editing of scale and coordinates for the system overall. The following dialog box appears:



The system's scale can be increased or decreased via the given factors or all coordinates for constrained modulus profile and foundation points be displaced by fixed amounts. It is also possible to alter all point coordinates by rotation.

9.11.9 "Language preferences" menu item

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

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

10 Index

A

B

C

D

E

F

G

H

I

L

M

N

O

P

R

S

T

U

V

W

Z