Modelling of plane and axis-symmetrical strain conditions (FEM)

GGU-ELASTIC

VERSION 5

Last revision: February 2019

Technical implementation, layout and sales: Civilserve GmbH, Steinfeld

Contents:

List of Figures and Tables:

1 Preface

The **GGU-ELASTIC** program allows modelling plane and axis-symmetrical strain conditions on the basis of Hooke’s law. The finite-element method with triangular elements is used to solve the differential equation.

The program system includes a powerful mesh generator and easy-to-use routines for comfortable evaluation of the modelling results (contours, 3D graphics, etc.).

It is not the purpose of this manual to give an introduction to finite element methods. For details of finite element methods you are referred to O. C. Zienkiewicz, "Methode der Finiten Elemente" (Finite Element Methods), published by Carl Hanser Verlag, Munich, Vienna, 1984.

Data input is in accordance with conventional WINDOWS operations and can therefore be learned almost entirely without the use of a manual. 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 and has been thoroughly tested (using analytical solutions and in comparison with other finite-element programs). No errors have been discovered. Nevertheless, a liability for completeness and correctness of the program system and the manual, and for damage resulting from any incompleteness, cannot be accepted.

2 Licence protection

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

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

3 Language selection

GGU-ELASTIC 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).

4 Starting the program

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

- File
- Info

By going to the "**File**" menu, a previously analysed system can be loaded by means of the "**Load**" menu item, or a new one created using "**New**". You then see eight menus in the menu bar after selecting your system:

- File
- FEM mesh
- Boundary
- System
- Graphics preferences
- Page size + margins
- Evaluation
- 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-ELASTIC** screen is not refreshed after every alteration.

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

5 Short description

As, from personal experience, the reading of manuals is a chore, there will now follow a short description of the main program functions. After studying this section you will be in a position to analyse a strain state using the finite-element-method. You can take program details from the further chapters.

As well as the short description, the last chapter of this manual contains a concrete example, with which program use is thoroughly described.

- Design the system that you wish to analyse.
- Start the **GGU-ELASTIC** program and select the menu item "**File/New**". Select the type of system.
- If necessary, fit the page coordinates to those of your system. For this, use the menu item "**Page size + margins/Manual resize (editor)**".
- Then select the menu item "**FEM mesh/Define nodes**".
- Click on the decisive nodes (points) of your system with the mouse. The points will be numbered. Alternatively, you can enter the system nodes in tabular form, using the menu item "**FEM mesh/Change (nodes)**" or import them via the Windows clipboard e.g. from an Excel table.
- If the nodes lie outside of the page coordinates, select the menu item "**Page size + margins/Auto-resize**".
- Then select the menu item "**FEM mesh/Manual mesh**" and combine the nodes, in groups of three, to triangular elements. In this way you create a rough structure for your system. Alternatively, you can have the program do this automatically, using the menu item "**FEM mesh/Automatic**".
- If you would like to edit the positions of mesh nodes, select the menu item "**FEM mesh/Change (nodes)**", "**FEM mesh/Move (nodes)**" or "**FEM mesh/Edit (nodes)**".
- If you would like to analyse a simple rectangular system, you can complete mesh generation in a few seconds using the menu item "**FEM mesh/Array**".
- If you would like to delete a triangular element, select the menu item "**FEM mesh/Manual mesh**" again and click on the three corner nodes of the appropriate element. Using this menu item , try double-clicking in a triangular element.
- If you would like to assign different material properties to different triangles, use the menu item "**Boundary/Individual materials**" or "**Boundary/(Materials) In section**" to enter varying material numbers for individual or multiple elements. In the menu item "**System/Material properties**" an input line appears for each material number.
- You can refresh the screen at any time, using the [**Esc**] or [**F2**] keys.
- You can create a finer structure for your system by selecting the menu items "**FEM mesh/Refine individually**", "**FEM mesh/(Refine) Section**" or "**FEM mesh/(Refine) All**".
- Even after refining your FEM mesh you can edit the system as you wish by using "**FEM mesh/Define nodes**", "**FEM mesh/Manual mesh**", etc.
- In order to get a numerically favourable FEM mesh, you should always select the menu item "**FEM mesh/Optimise**" and then select the button "**Diagonals**".
- For demonstration purposes you can create acute angled, and thus numerically unfavourable, triangular elements with the menu item "**FEM mesh/Move (nodes)**". Then select the menu item "**FEM mesh/Optimise**" ("**Topology**" button) and follow the effects on the screen.

- Define the decisive displacement boundary conditions for your system using, e.g., the menu item "**Boundary/Individual displacement BC**" or "**Boundary/(Displacement BC) In section**".
- Define the decisive force boundary conditions for your system using, e.g., the menu items "**Boundary/Point loads**" or "**Boundary/Line loads**".
- If necessary, edit the material properties using the menu item "**System/Material properties**".
- If beam elements (e.g. piles or anchors) are present in the system, define the position of these elements via the menu item "**Boundary/Beams**", by defining a ***section*** along the desired nodes. After pressing the [**Return**] key you can assign the thus defined beams a material number. The beam stiffness' (EJ and EF) can be edited using "**System/Material (beams)**".
- Simple control of the defined boundary conditions is possible using the menu item "**Boundary/Check**".
- When you have finished FEM mesh generation, select the menu item "**System/Analyse**", and start the analysis. Before analysis begins a bandwidth optimisation will be carried out, if necessary, in order to achieve a numerically favourably configured equation system.
- After completion of analysis you can, if wished, print the results as an output table or save them as a file. In general though, this type of result presentation is less than satisfactory.
- Instead, go immediately to the "**Evaluation**" menu. Here you have a variety of evaluation possibilities. The menu item "**Evaluation/Coloured contours**" is especially impressive, or the menu item "**Evaluation/3D array contours**". The dialog boxes which then appear can almost always be left using "**OK**", without further changes having to be made. The program will usually suggest sensible input. The "**Determine extreme values …**" button should be clicked once however, otherwise an error message will appear, with correction suggestions.
- If the connected printer is a colour printer and is correctly installed in WINDOWS, you can create colour output by selecting the menu item "**File/Print and export**" and then selecting the "**Printer**" button in the dialog box which appears. For black and white printers, grey scales will be used.
- At the end of the manual, program use is described in detail using a concrete example.

When carrying out analyses, please remember that all finite-element-methods are approximation methods. The quality of the approximation increases with increasing mesh density. In the current version, systems with a maximum of 225,000 triangular elements and 225,000 nodes can be analysed.

Several examples from the literature are saved on disk (Zienkiewicz and Schwarz). Recalculation shows almost complete agreement.

The short description shows that only a few menu items need be selected for analysis. All further menu items are mainly for data saving, layout and, if necessary, further evaluation of analyses. A description will follow in the further chapters.

6 Worked example

6.1 Example system

Following, analysis will be shown on a concrete example. Stress distribution below a strip foundation is to be analysed.

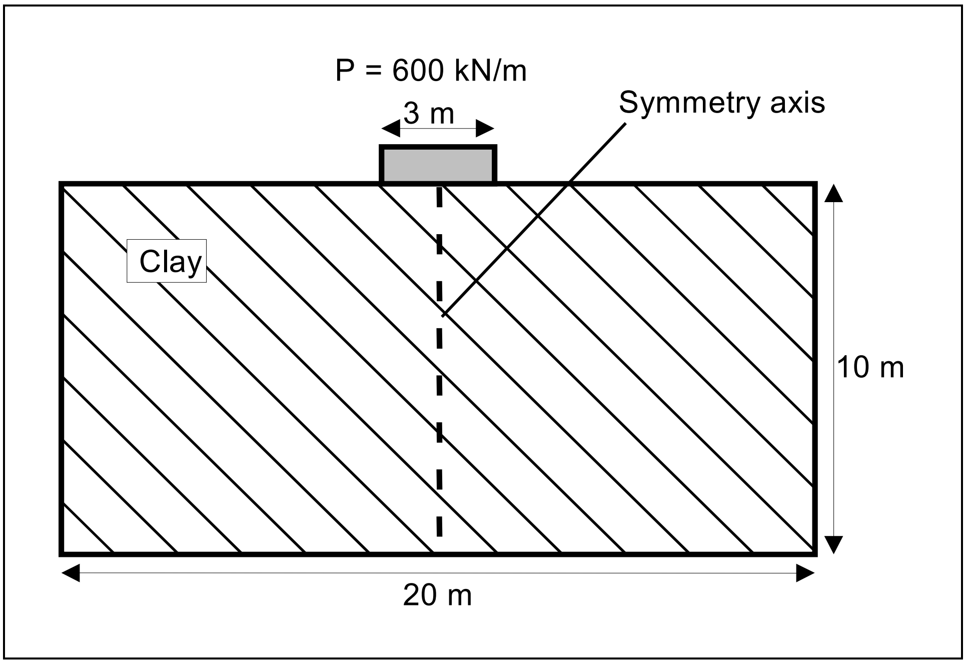


Figure 1 Example system

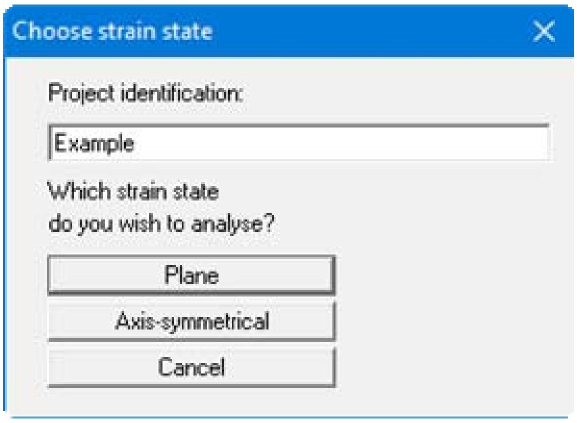
The following characteristics are given:

- Constrained modulus E_S = 7.5 MN/m²
- Poisson's ratio ν = 0,0
- Friction angle φ = 25°
- Cohesion c = 30 kN/m²

Unit weight input is not necessary as the subsurface system is already settled due to the soil self-weight. The unit weight is therefore set to "0". A foundation pressure of 200 kN/m² results from the load P and the foundation width.

6.2 Step 1: Program preparation

Start the program and select the menu item "**File/New**".

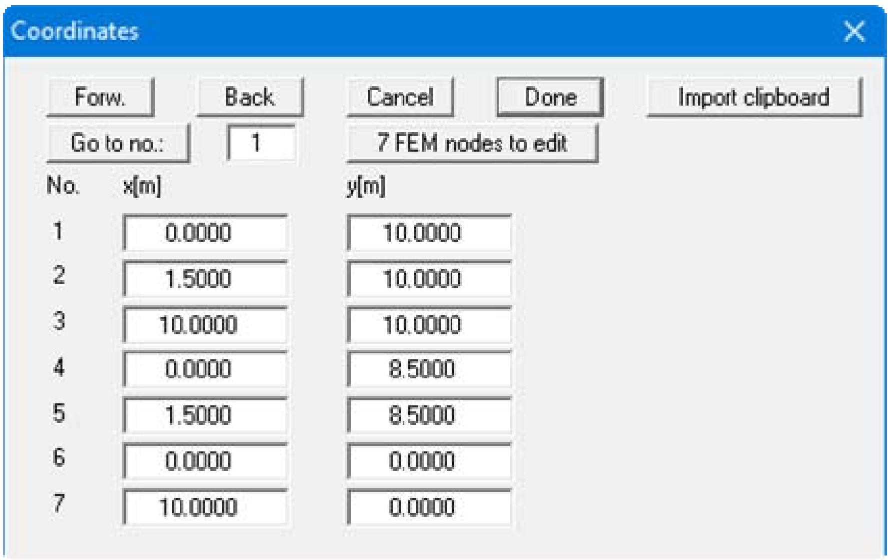


Select the "**Plane**" button from the dialog box.

6.3 Step 2: System input

The problem is symmetrical, so we need only consider one half. We will choose the right half. The coordinate system should be selected such that the foundation centre is at x = 0.0 m. The foundation base has a y-value of 10.0 m.

The system can be described with a few distinctive points, so input can be carried out easily using a table. Select the menu item "**FEM mesh/Change (nodes)**" and then the "**Via a table**" button. Change the number of nodes to 7 and enter the values from the following dialog box.



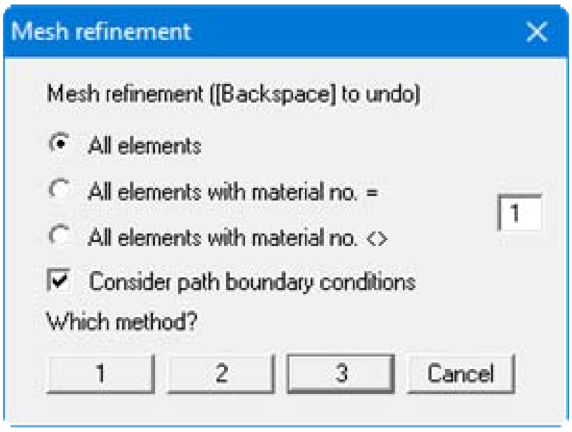
In principle, 5 points would have been sufficient to describe the system:

- four corner points
- right foundation edge

However, two additional points have been added in order to get a slightly tighter mesh in the foundation area.

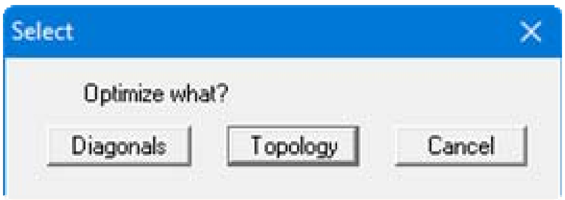
Select the menu item "**FEM mesh/Automatic**". The program connects the seven points to a FEM mesh. Then go to the menu item "**Page size + margins/Auto-resize**" or click the [**F9**] function key to get a screen-filling system presentation.

Select the menu item "**FEM mesh/(Refine) All**" and press button "**3**".



Repeat the mesh refinement. You should now have a system with 96 triangles and 61 nodes. The current numbers are displayed in "**System/Info**".

Now select the menu item "**FEM mesh/Optimise**".



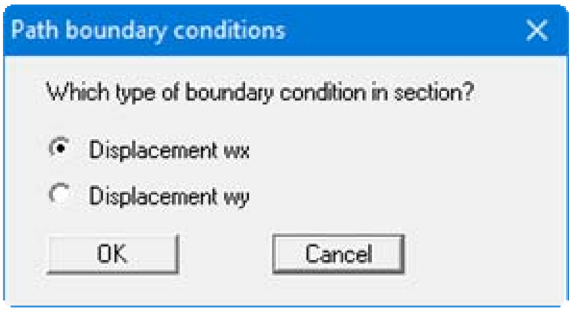
Press the "**Topology**" button to get a numerically favourable mesh.

6.4 Step 3: Boundary conditions

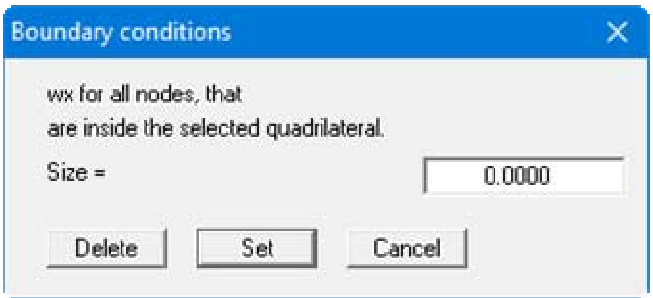
The following boundary conditions are given:

- left system boundary immovable in x-direction,
- right system boundary immovable in x-direction,
- lower system boundary immovable in x and y-direction,
- line load of 200 kN/m/m below the foundation.

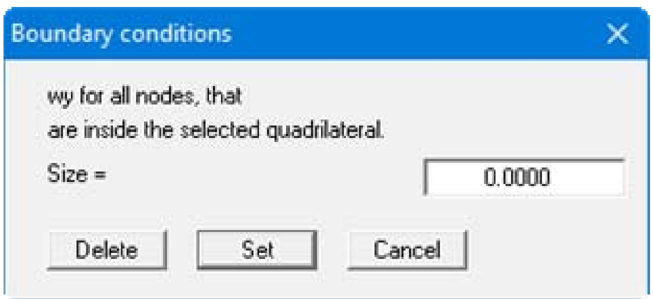
Select the menu item "**Boundary/(Displacement BC) In section**".



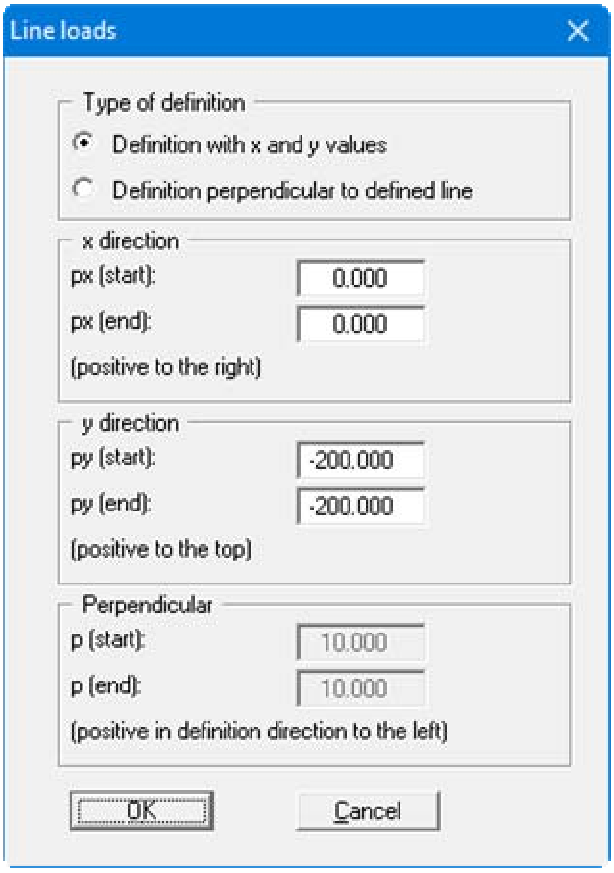
Conform the "**Displacement wx**" with "**OK**". Trace the left boundary nodes and set the displacement in x-direction to "**0**".



Repeat this for the lower and the right system boundaries. Select the menu item "**Boundary/(Displacement BC) In section**" and activate the check box for displacement boundary conditions in y-direction. Assign the lower system boundary a displacement in y-direction of "**0**".



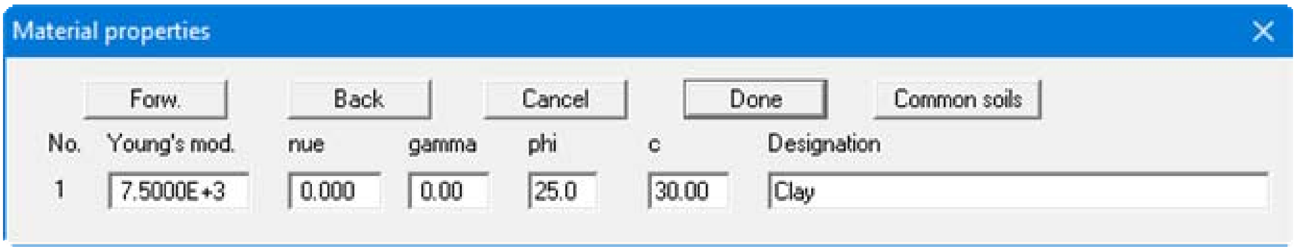
Select the menu item "**Boundary/Line loads**" and click on the foundation centre and the right foundation edge. After pressing the [**Return**] key, define the foundation stress at 200 kN/m² via definition with x and y values.



Select the "**FEM mesh/(Refine) All**" menu item once again, to get a finer FEM mesh. You should now have a system with 384 triangles and 217 nodes (see "**System/Info**" menu item).

6.5 Step 4: Material properties

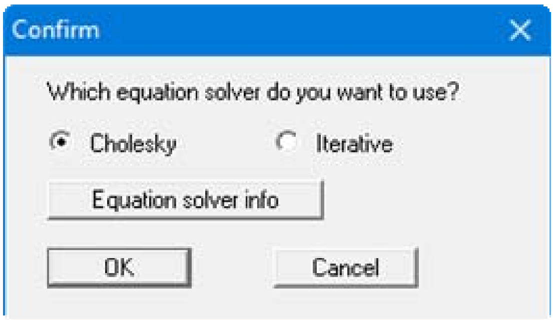
Select the menu item "**System/Material properties**" and enter the following values:



In the procedure as described above, all elements are assigned the material number 1 during automatic mesh generation. If you have defined several materials when defining the triangular grid (for example, using the "**Boundary/Individual materials**" menu item), the dialog box above will correspondingly contain several lines for the individual materials.

6.6 Step 5: Analyse system

Select the menu item "**System/Analyse**".



Now select "**Cholesky**" as the equation solver and start the analysis using the "**Start**" button in the subsequent dialog box. Depending on the power of your computer, you will be informed after a certain time that analysis is complete.

6.7 Step 6: Evaluate

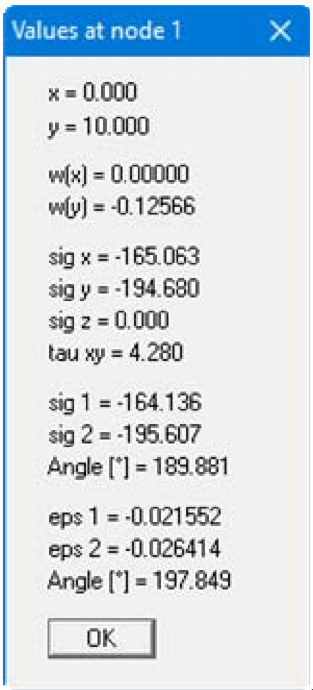
First, the results are to be compared to exact solutions. Using the relationships from the "Grundbautaschenbuch (Volume 1; "Stress determination")" the exact values for this case can be analysed.

Table 1 Settlements [cm] for worked example

x [m]	Results GGU-ELASTIC	exact	Difference [cm]	Difference [%]
0.0000	12.6	12.2	0.4	3.3
0.1875	12.5	12.2	0.3	2.5
0.3750	12.4	12.1	0.3	2.5
0.5625	12.2	11.9	0.3	2.5
0.7500	11.9	11.6	0.3	2.6
0.9375	11.4	11.2	0.1	0.9
1.1250	10.8	10.7	0.1	0.1
1.3125	9.8	10.0	-0.2	-2.0
1.5000	8.8	8.8	0.0	0

The results deviate by a maximum of 3.3 % from the exact solution.

If you are interested, as in this case, only in individual values at certain nodes, this can be achieved most simply via the "**Evaluation/Individual values**" menu item. After selecting this menu item, left-click the required node. The dialog box below shows the result for the first row of Table 1 (foundation centre at x = 0.0 m).

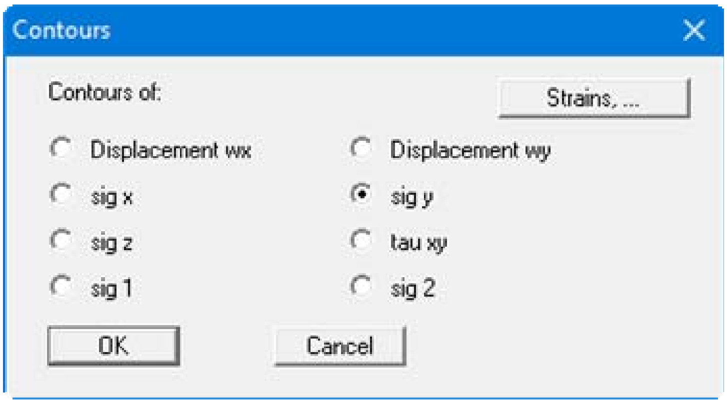


,

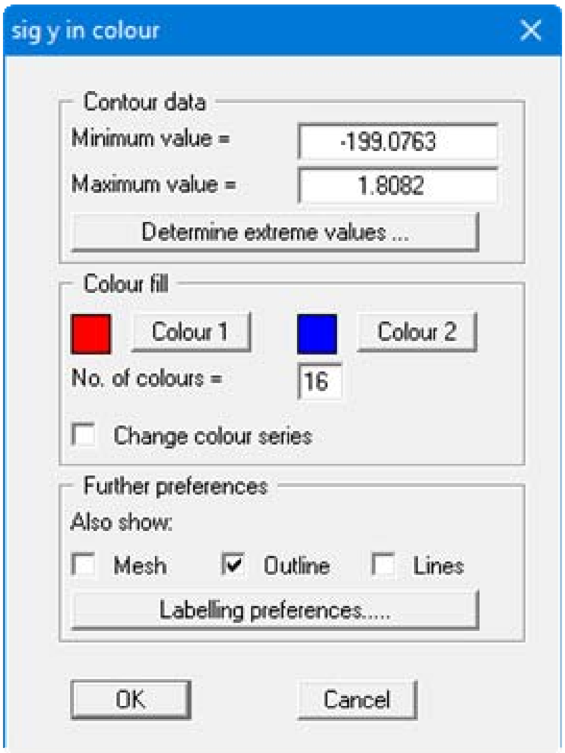
6.8 Step 7: Display the stress bulb

Finally, the stress bulb (lines of equal stress in y-direction) is to be displayed.

For this, select the menu item "**Evaluation/Coloured (contours)**".



Activate the "**sig y**" check box and confirm with "**OK**". The following dialog box will be shown.



Press the "**Determine extreme values**" button once and then the "**OK**" button.

If the colour bar at the right is drawn over the right edge of the sheet, go to "**Page size + margins/Page size and margins**" menu item and select a larger "**Right plot margin**" (see Section 8.6.4). If you would like to print the stress bulb, go to the menu item "**File/Print and export**".

In a similar manner you can create contours of displacements, strains, etc. (menu item "**Evaluation/Normal contours**" etc.), or determine bearing forces (menu item "**Evaluation/Supports**"). You can create circle graphics in which the appropriate state variables are shown as circles of different sizes (value-dependent).

You can freely label the graphics using "**Graphics preferences/Mini-CAD toolbar**" or create a header for your output sheet using "**Graphics preferences/Header toolbar**". Please read the **Mini-CAD** manual (see Section 8.5.5).

7 Theoretical principles

7.1 General

An analytical solution is only possible for simple systems. When modelling complicated systems, numerical solutions are required. In the main

- finite-difference-methods (FDM) and
- finite-element-methods (FEM)

are used. Using finite methods, the whole area is subdivided into a large amount of smaller (finite) areas (elements). Generally, using FEM, triangles are selected for these areas. Within these triangles simple, generally linear, approximation functions are used. The actual, complicated, whole solution is pieced together like a mosaic from the many simple partial solutions. Equation systems result, in which the number of variables corresponds to the number of system nodes. With the finite-difference-method one generally has only the possibility of defining a discrete whole using rectangular partial-areas. In contrast to FEM therefore, FDM is a lot less flexible when it comes to adjusting to complicated boundary structures. Also, in some areas, mesh refinement is not as easy. Further, the resulting equation systems are numerically more stable for FEM. The main advantage in FDM is in the theoretically simpler mathematical relationships, which generally will be of little interest to the program user. The **GGU-ELASTIC** program uses the finite-element-method.

When using the application please remember that all finite-element or finite-difference-methods are approximation methods. The quality of the approximation to the actual solution increases with an increase in mesh refinement. You should pay attention to the fact that the mesh should be finer for areas in which the main forces act (e.g. point loads). The type of triangle used also exerts a certain influence. Optimum conditions are achieved with equilateral triangles. You can get an overview of the solution quality by analysing the same system with a coarser and a finer mesh subdivision, and comparing the deviation of the two results.

The following general notes to the **GGU-ELASTIC** program are important:

- Triangular elements are used.
- Hooke’s Law is valid.

Analytical solutions for the differential equation for plane and axis-symmetrical strain conditions only exist for a few special cases, so that for problems of daily design practice (with variously distributed loads, free, fixed or supported boundaries etc.) numerical methods must be relied upon.

The differential equation is solved by the program using finite-element-methods. Triangular elements are used for this. Some simple assumptions are made for these triangular elements with references to displacements. In the present case, a linear displacement assumption is used, which is described in Zienkiewicz (Carl-Hanser-Verlag, 1984, Chapters 4 and 5). The selected assumption leads to equation systems, whose number of variables correspond to twice the number of system nodes. The complete solution is put together like a mosaic from the many partial solutions of the triangular elements. It is clear that with increasing refinement of the finite-element mesh, the quality of the solution also increases.

The stresses are determined by numerical differentiation of the strains. As a linear displacement assumption was selected, the stresses are constant for each element. To compensate for this, the program follows a suggestion of Zienkiewicz' thus:

> For each element node the stresses from the neighbouring elements are added and then divided by the number of neighbouring elements. In this way, the stress course can be better presented. Naturally, the results at the boundary nodes are not quite as exact. Further, the approximation of stresses in the region of element nodes belonging to elements with different material types can be worsened by doing this. If the stresses at such ***boundary nodes*** are of great interest, an improvement can be achieved by refining the FEM mesh in these areas.

The quality of the calculated strains is, generally, excellent. If you are only interested in strains, you need not worry about the previous explanations.

Continue to remember that all finite-element-methods are approximation methods. The quality of the approximation increases with increasing mesh density.

> The case of a freely supported boundary is automatically considered by the finite-element-method. Valid is, that all system boundaries or partial system boundaries, which have no force or displacement boundary conditions, are automatically freely supported.
>
> In finite-element theory this type of boundary condition is also known as a natural boundary condition.

7.2 Signs and designations

The following sign rules are valid:

- displacement wx and wy e.g. in [m] positive in positive axis direction
- point loads Px and Py e.g. in [kN] positive in positive axis direction.
- stresses positive when tensile
- strains positive when tensile

Further designations:

- x, y = coordinates
- ν = Poisson's ratio [-]
- E = Young's modulus e.g. in [kN/m²]

Designation of stresses for plane strain conditions:

- sig x = stress in x - direction
- sig y = stress in y - direction
- sig z = stress in z – direction (normal to plane of observation)
- sig 1 and sig 2 = principal stresses
- tau xy = shear stress in plane of observation

Designation of stresses for axis-symmetrical strain conditions:

For axis-symmetrical strain conditions the horizontal axis is r and the vertical axis is z. The rotational axis is at r = 0.0 m (of course).

- sig r = stress in r – direction (horizontal)
- sig z = stress in z – direction (vertical)
- sig t = stress in t – direction (normal to plane of observation)
- sig 1 and sig 2 = principal stresses
- tau rz = shear stress in plane of observation

The program works with true dimensions. This means that you can freely choose the dimensions for force and length. If, for example, you decide on [kN] and [m], all input must be in these dimensions (e.g. then, Young’s modulus in kN/m²).

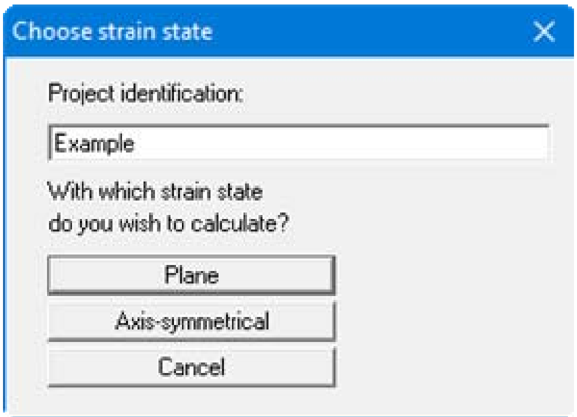
The friction angle should always be given in sexagesimal degrees (right angle = 90°).

8 Description of menu items

8.1 File menu

8.1.1 "New" menu item

After selecting the "**New**" menu item , the type of system to be created must be determined in the following dialog box. You can enter a dataset description ("**Project identification**"), which will then be used in the ***General legend*** (see Section 8.5.8).



If a FEM mesh has already been entered and you would like to switch from a plane to an axis-symmetrical system (or vice-versa) you can utilise the existing mesh, after confirming a query.

8.1.2 "Load" menu item

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

8.1.3 "Save" menu item

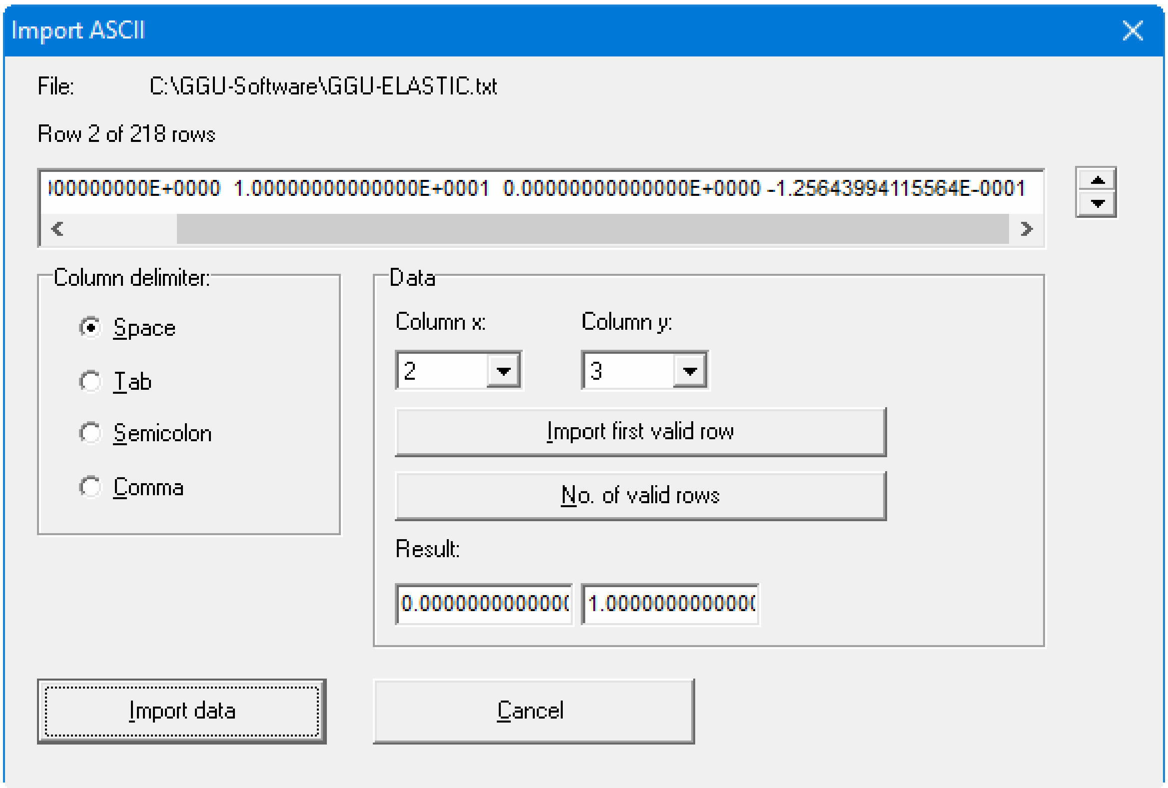
You can save data entered or edited during program use to a file, in order to have them available at a later date, or to archive them. The data is saved without prompting with the name of the current file. Loading again later creates exactly the same presentation as was present at the time of saving.

8.1.4 "Save as" menu item

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

8.1.5 "Import ASCII file" menu item

If the coordinates of the FEM mesh nodes are available in ASCII file format, they can be imported into the program. Each row of the file must contain the x- and y-value of a node. Decimal fractions must use a point, not a comma. When importing the ASCII file you must specify the columns containing the x- and the y-values.



The current row of the ASCII file is shown at the top. You can navigate through the file using the arrow buttons on the right. If all the information is correct, the result for the row is shown in the box below the column. Otherwise, an error message appears.

You may need to change the column delimiter.

If the file contains invalid as well as valid rows, these will simply be skipped during the subsequent import. After the "Import data" button has been pressed and the data successfully imported, you will see a message box with the number of points and character strings imported. The imported coordinates can then be processed to form an FEM mesh.

8.1.6 "Export ASCII file" menu item

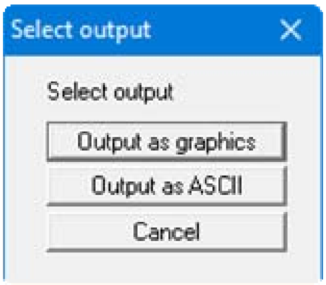
If an FEM mesh has been generated, the node coordinates can be saved to an ASCII file, allowing them to be imported into other programs where required.

8.1.7 "Print output table" menu item

8.1.7.1 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.

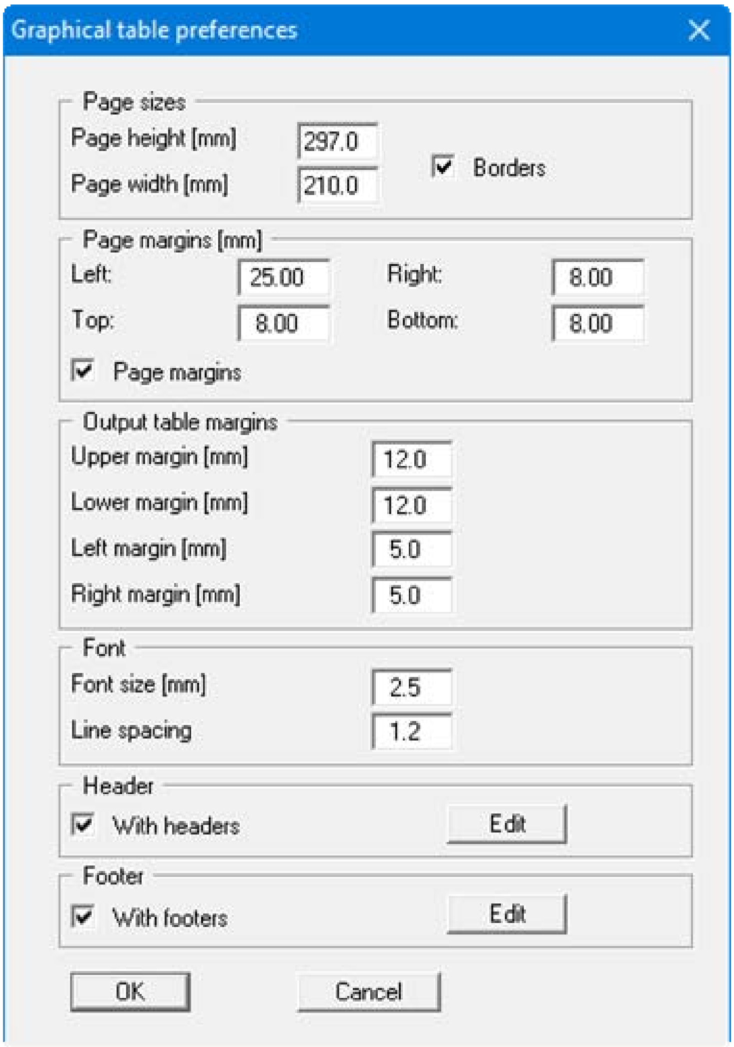
You have the option of designing and printing the output table as an annex to your report within the **GGU-ELASTIC** program. To do this, select "**Output as graphics**" from the following options.



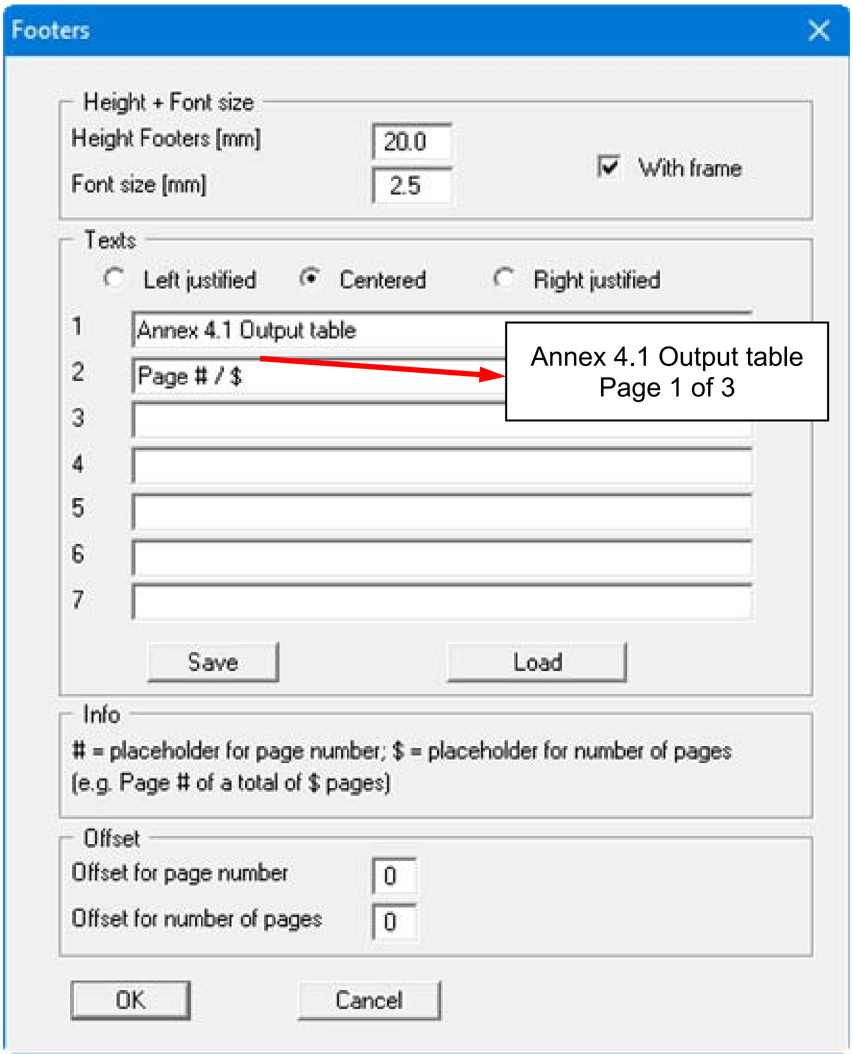
If you prefer to easily print or process the data in a different application, you can send them directly to the printer or save them to a file using the "**Output as ASCII**" button.

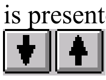
8.1.7.2 Button "Output as graphics"

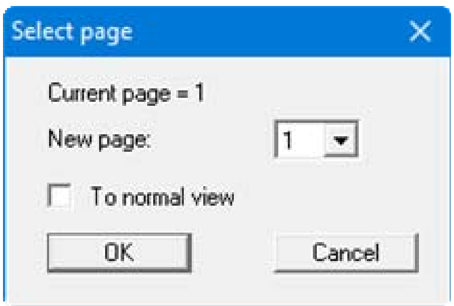
If you selected the "**Output as graphics**" button in the previous dialog box a further dialog box, in which you can define further preferences for result presentation.



You can define the desired layout for the output table in various areas of the dialog box. If you need to add a header or footer (e.g. for page numbering), activate the appropriate check boxes "**With headers**" and/or "**With footers**" and click on the "**Edit**" button. You can then edit as required in a further dialog box.

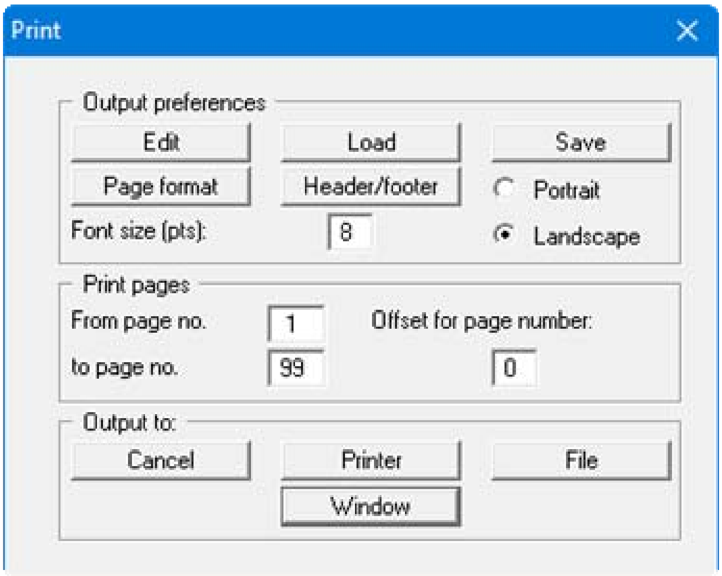


Automatic pagination can also be employed here if you work with the placeholders as described. After exiting the dialog boxes using "**OK**" you will see a further dialog box in which you can select the parameters to be used in the output table. If you click the "**Start**" button the output table is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



8.1.7.3 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-ELASTIC.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.

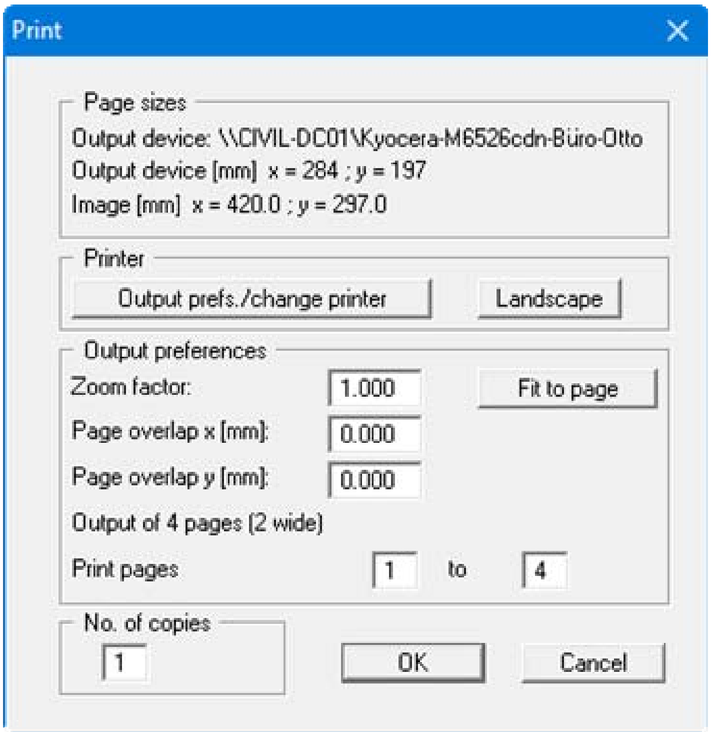
8.1.8 "Output preferences" menu item

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

8.1.9 "Print and export" menu item

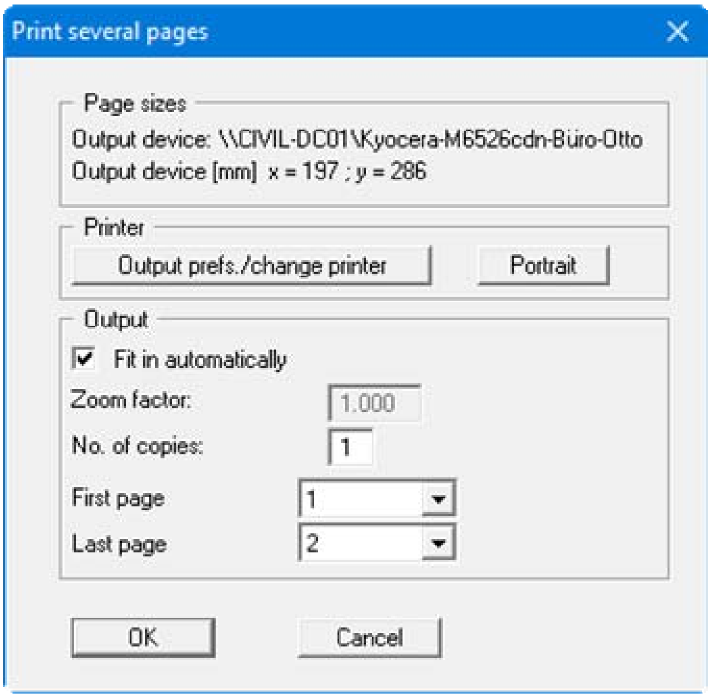
You can select your output format in a dialog box. You have the following possibilities:

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

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

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

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

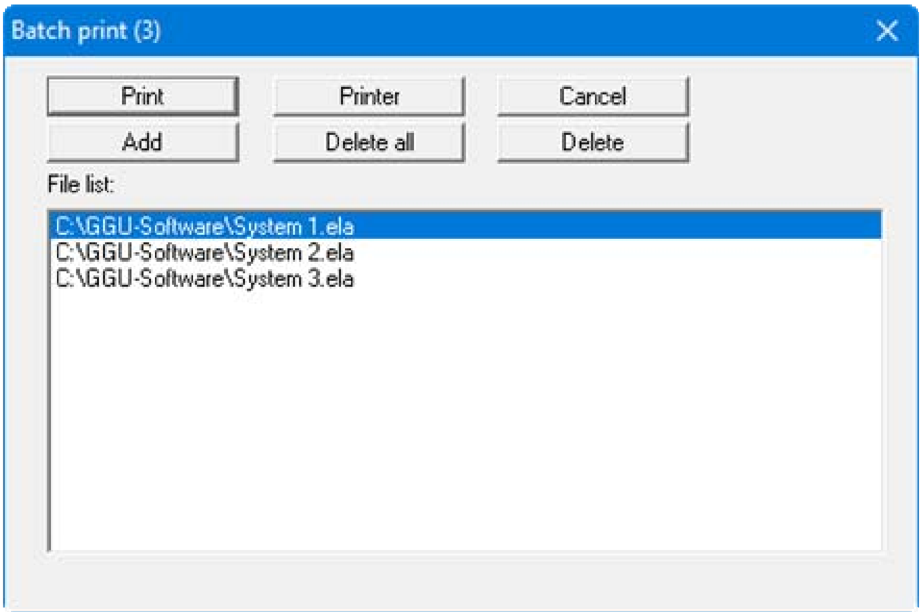
If you select the "**Copy/print area**" tool  from the toolbar, you can copy parts of the graphics to the clipboard or save them to an EMF file. Alternatively you can send the marked area directly to your printer (see "**Tips and tricks**", Section 9.3).

Using the "**Mini-CAD**" program module you can also import EMF files generated using other GGU applications into your graphics (see Section 8.5.5).

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

8.1.10 "Batch print" menu item

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



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

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

8.1.11 "Exit" menu item

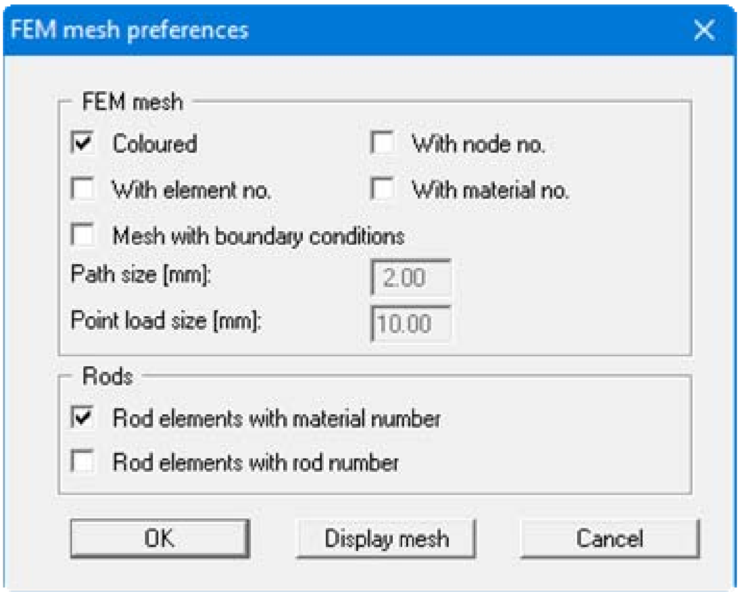
After a confirmation prompt, you can quit the program.

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

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

8.2.1 "Preferences" menu item

Using this menu item you can define the appearance of the FEM mesh on the screen. The element number and material number cannot be displayed simultaneously.



The settings are only adopted after closing the box using "**OK**". The "**Display mesh**" button produces a direct representation of the FEM mesh using the selected preferences.

8.2.2 "FEM mesh" menu item

After going to this menu item the FEM mesh is displayed as defined in "**FEM mesh/Preferences**".

If the FEM mesh representation does not fill the screen, point to the "**Auto-resize**" menu item in the "**Page size + margins**" menu or press [**F9**].

8.2.3 "Outline" menu item

After selecting this menu item the outline of the various soils in the FEM system, as defined in "**FEM mesh/Preferences**", will be displayed.

8.2.4 "Define nodes" menu item

With the left mouse button you can set a new node, and with the right mouse button you can delete a previously set node. The menu item can also be reached with the [**F3**] function key.

> If you do not want to click every node individually, it is possible to define several nodes at once using the menu item "FEM mesh/Edit (nodes)" (Section 8.2.7) or "FEM mesh/Array" (Section 8.2.8)

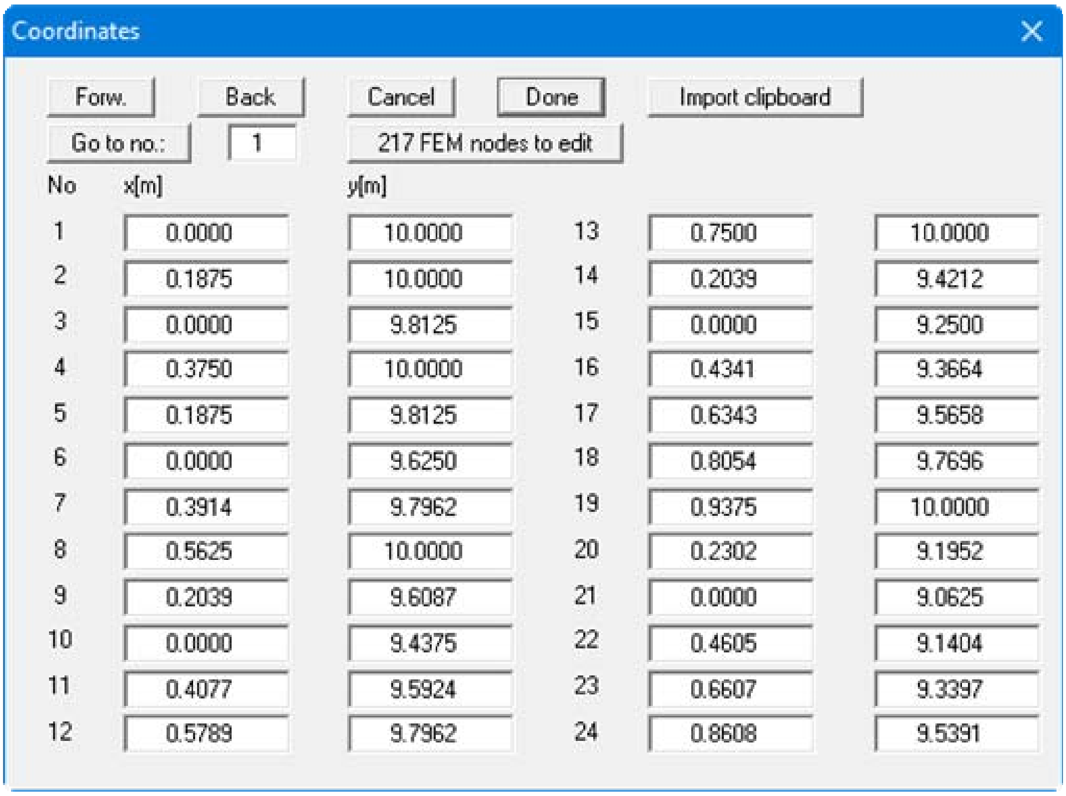
When setting new nodes, the current x- and y-coordinates of the mouse are shown in the status bar at the bottom of the window. If you have access to a scanner you can scan in the system to be processed and save it e.g. as a bitmap file (extension: "**.bmp**"). This bitmap can be displayed on the screen using the **Mini-CAD** program module (see the "**Mini-CAD**" manual). This greatly simplifies input of the principal system nodes.

Alternatively, you can also import a DXF file using **Mini-CAD** (see "**Mini-CAD**" manual). This can contain the system outline, for example. If **Mini-CAD** data are already present, the initial dialog box provides a check box with the option of locking on to **Mini-CAD** lines. If you activate this check box the mouse cursor appears as a rectangle with cross-hairs. If the end point of a **Mini-CAD** line is located within this rectangle the program will lock on to this point precisely; if a number of points are located within the rectangle, it will lock on to the one nearest the centre of the cross-hairs.

8.2.5 "Change (nodes)" menu item

Using this menu item you can edit the FEM nodes previously entered with the mouse or, alternatively, enter new node coordinates for a new system. Two options are available for this:

- **"Via a table"**
  You can edit the coordinates of existing nodes in a dialog box or, alternatively, enter the coordinates of new nodes.

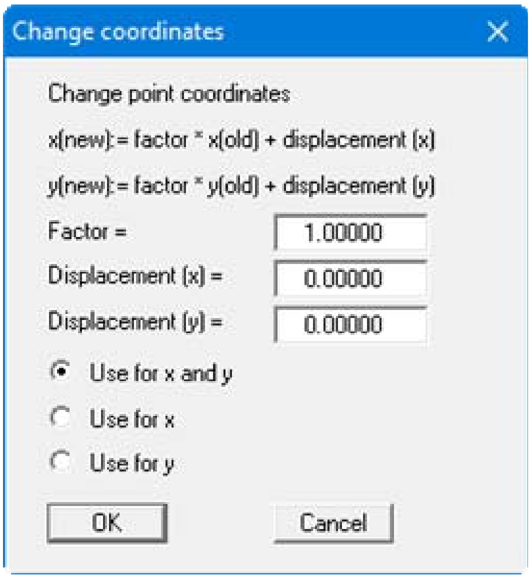


If you need to edit the current number of nodes click the "**x FEM nodes to edit**" button and enter the new number of nodes. You can navigate through the table using "**Forw.**" and "**Back**".

> If you set the number of nodes to **0** the FEM incidence table is deleted.

It is even easier to import node coordinates via the Windows clipboard. For example, if the x-/y-coordinates of the FEM mesh nodes are available in an Excel table, it is possible to copy the two columns containing the data into the clipboard ("*Edit/Copy*") and then to paste them into the dialog box above by pressing "**Import clipboard**".

- **"Via equation"**
  If you have entered the coordinates using the wrong scale, for instance, you can correct this using this menu item.



  You also have the option of applying the factor for a specific direction only. Activate the required option buttons.

8.2.6 "Move (nodes)" menu item

After selecting this function, the defined FEM system is displayed with the FEM elements. Each node can be moved with the left mouse button pressed. The coordinates of the current node are displayed in the status bar. The last node movement can be undone using the [**Backspace**] key.

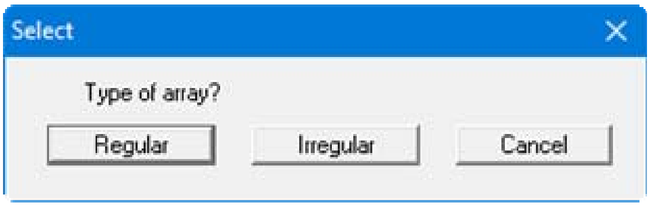
8.2.7 "Edit (nodes)" menu item

By double-clicking a node using the left mouse button a dialog box appears allowing the coordinates to be edited via the keyboard.

8.2.8 "Array" menu item

8.2.8.1 Select type of array

If you want to enter several nodes simultaneously in a specific array, you can achieve this easily using this menu item. You can also use the function to extend an existing system. First, decide whether the new nodes are to generated using a regular or an irregular array.

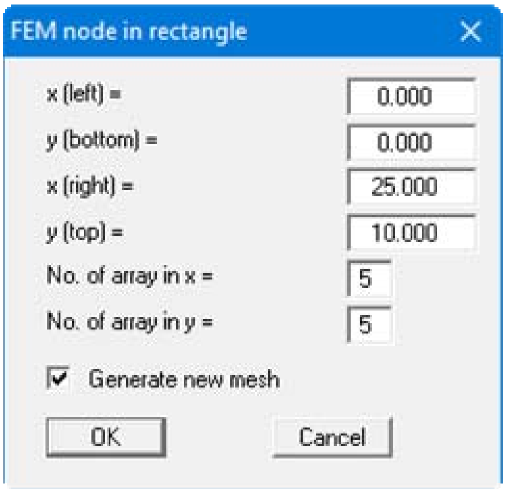


8.2.8.2 Button "Regular"

The regular array allows the nodes to be defined in a number of ways:

- "**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.

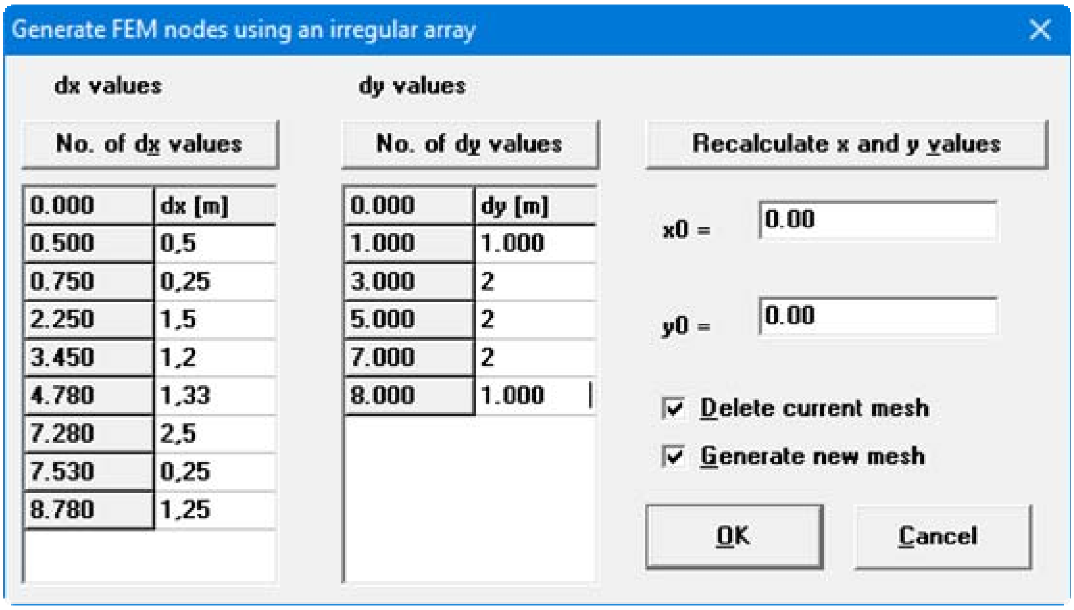


Enter the corner points of the array and the number of subdivisions. If the "**Generate new mesh**" check box is activated, all existing nodes are deleted and then the user-defined FEM nodes in the dialog box are generated with the appropriate FEM mesh.

> If you would like to build a system consisting of several rectangles, the "**Generate new mesh**" check box must be deactivated. The existing FEM nodes and the associated mesh are then retained.

8.2.8.3 Button "Irregular"

In contrast to the regular array procedure, where the generated nodes are evenly spaced within the generated rows, the spacing can be varied using the irregular array. This can be done in the following dialog box.



The array numbers can be defined using the "**No. of dx values**" and "**No. of dy values**" buttons. Enter the array spacing in "**dx**" and "**dy**". If you press the "**Recalculate x and y values**" button the first column to the left of "**dx**" and "**dy**" are recalculated. These are absolute values for the array. To determine these absolute values, the program uses the array origins, which you specify using "**x0**" and "**y0**".

The program can generate an FEM mesh from the newly generated FEM nodes if the "**Generate new mesh**" check box is activated. If a mesh already exists, it can be deleted prior to generating the new one by activating the "**Delete current mesh**" check box.

8.2.9 "Manual mesh" menu item

After mesh node input, a finite-element mesh is defined with this menu item. Three nodes must be clicked using the left mouse button. Once the three nodes have been selected a box appears for defining the material number of the FEM element. This menu item can also be reached using the [**F4**] function key.

If you click three nodes of an existing FEM element, you will see a dialog box that you can use to delete the element or to change the material number. In order to change the material no. of an FEM element, you can also open the editing box by double-clicking over the element.

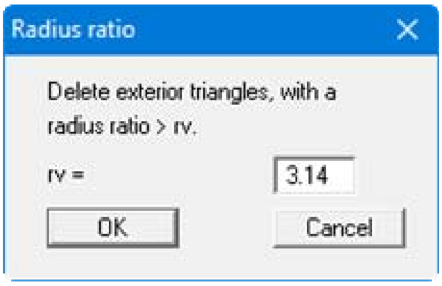
8.2.10 "Automatic" menu item

After entering the mesh nodes automatic mesh generation can be carried out using this menu item (Delauney triangulation). If an FEM mesh already exists it can be either deleted or supplemented. Under certain circumstances, ***air holes*** (incompletely filled FEM mesh) may occur in an existing FEM mesh being supplemented if this mesh was not generated by means of Delauney triangulation. These regions will require manual post-processing or re-triangulation of the complete FEM mesh. All newly generated triangles are assigned the material number **1**.

8.2.11 "Round off" menu item

During Delauney triangulation a triangular 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 FEM mesh using this menu item.

The program first draws the maximum radius ratio of the most unfavourable triangle and displays the data in a message box. You then see a dialog box, allowing you to define a maximum radius ratio above which all triangles with greater values are deleted.



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). In the example above, all external triangles with a radius ratio greater than 3.14 will be removed.

In order to avoid ***interpolation holes*** in the triangle system only triangles at the boundaries are deleted.

8.2.12 "Delete" menu item

This menu item allows you to 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.

When this menu item is selected, the mesh is displayed without node labelling. If you have deleted the entire mesh, you will also no longer see the nodes. Simply click "**FEM mesh/FEM mesh**" and the nodes will be displayed.

8.2.13 "Optimise" menu item

You first select in a dialog box whether the diagonals or the topology should be optimised.

- "**Diagonals**" button
  Optimisation of diagonals is implemented in order to create a numerically favourable FEM mesh, i.e. where possible, equilateral triangles. The effect of the optimisation of diagonals can best be seen using an example:

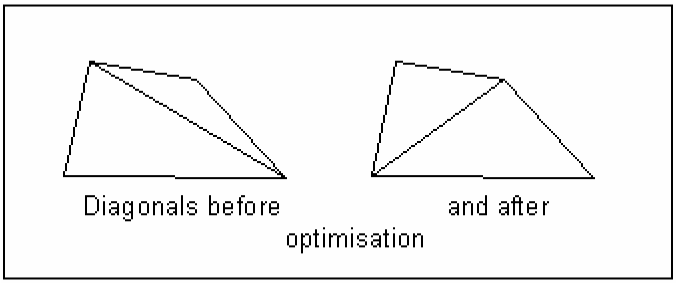


Figure 1 Optimisation of diagonals

> If an existing ***unfavourable*** diagonal cuts two different material areas, <u>no</u> optimisation takes place, because this would alter the system.

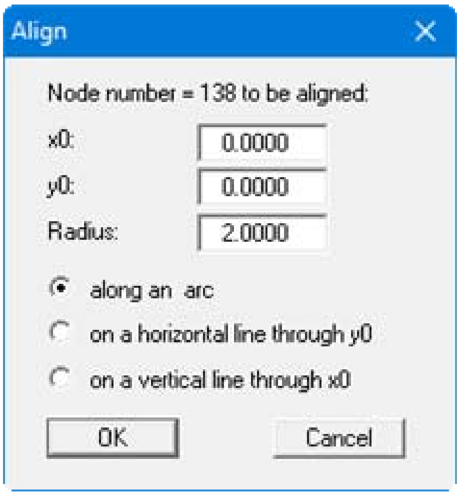
- "**Topology**" button
  This program routine displaces the triangular element nodes so that, where possible, equilateral triangles are created. Equilateral triangles have especially favourable numerical properties. Because the displacement of system boundaries and element boundaries with neighbouring elements consisting of different soils does not make much sense, these element boundaries are fixed from the outset. Nodes with defined water level boundary conditions also remain unaltered.

  Optimisation of the FEM mesh can be followed on the screen by setting the "**With graphics**" check box. The optimisation routine can be aborted at any time by pressing the right mouse button.

8.2.14 "Align" menu item

You can have the mesh aligned around circles or along lines. First, decide which criterion the program should follow:

- "**Nodes**" button
  You can select individual nodes and have them aligned according to your specifications.

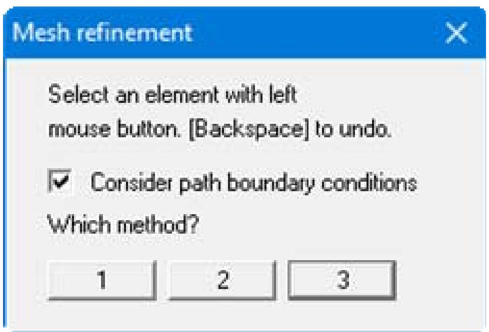


  Here, you have the option of aligning the selected nodes along an arc, on a horizontal line through y0 or on a vertical line through x0 with the spacing entered in the appropriate input box.

- "**Line**" button
  First, click several nodes to be connected by a line and confirm the selection using the [**Return**] key. You will then see the same dialog box as for "**Nodes**", only without information on the selected node numbers. After entering and selecting the desired type, the selected nodes will be aligned.

8.2.15 "Refine individually" menu item

FEM mesh elements can be selected for refinement using the following menu item.



Upon activating the "**Consider displacement boundary conditions**" check box, new nodes located immediately between two nodes with displacement boundary conditions will be assigned the average of the two values. This procedure is not unequivocal when applied to ***force boundary conditions*** and can lead to misunderstandings. Force boundary conditions are therefore <u>not</u> refined in the course of mesh refinement.

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

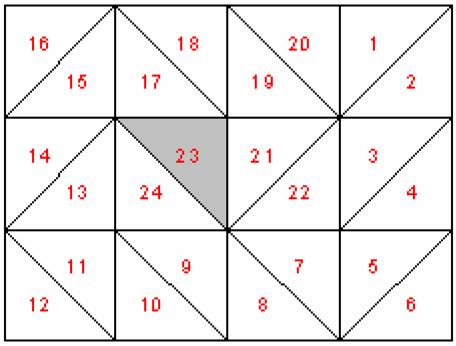


Figure 2 FEM mesh for refinement demonstration

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

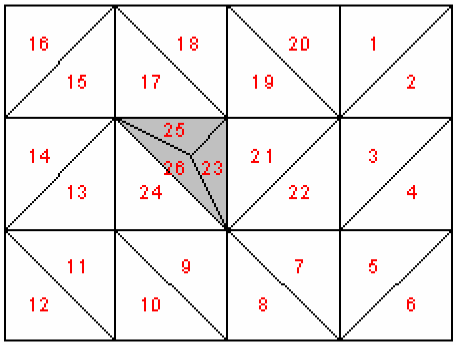


Figure 3 FEM mesh after refinement using Method 1

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

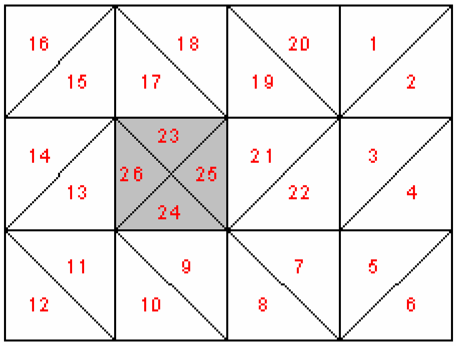


Figure 4 FEM mesh after refinement using Method 2

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

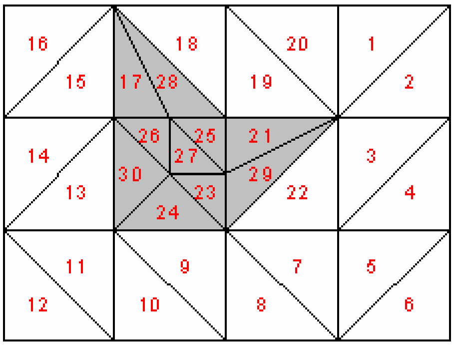


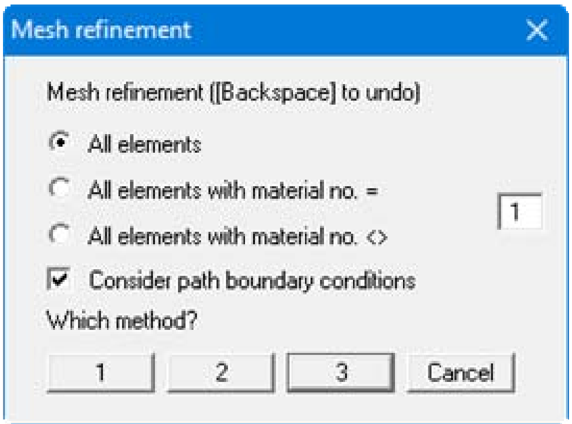
Figure 5 FEM mesh after refinement using Method 3

8.2.16 "(Refine) Section" menu item

A number of elements previously enveloped in a polygon can be refined using this menu item or, alternatively, pressing [**F6**]. Displacement boundary conditions can be taken into consideration (see description of the refinement methods in the "**FEM mesh/Refine individually**" menu item, Section 8.2.15).

8.2.17 "(Refine) All" menu item

The following dialog box appears after selecting this menu item or, alternatively, pressing [**F7**]:



Either all elements or only element with certain material numbers can be refined. Here, too, potential boundary conditions can be taken into consideration (see description of the refinement methods in the "**FEM mesh/Refine individually**" menu item, Section 8.2.15).

8.2.18 "Save/load mesh"

You can save your mesh in a "**.ggu_ntz**" file using this menu item, or import a previously saved mesh.

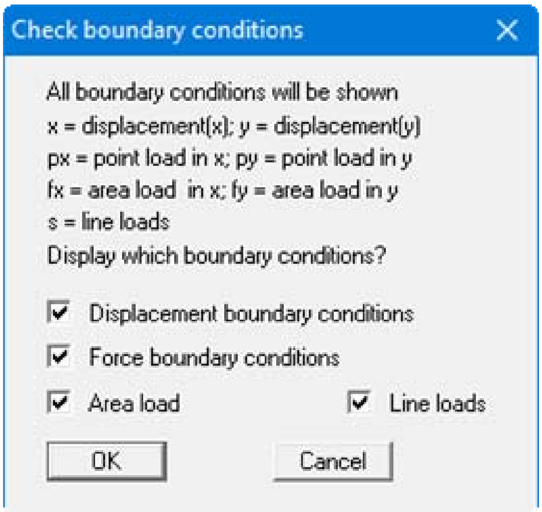
8.3 Boundary menu

8.3.1 "Preferences" menu item

This menu item allows you to specify the boundary condition display format on the screen. You can select visualisation using either symbols or numbers. You can specify the desired number formats separately for displacement and force boundary conditions. In addition, you can specify the font sizes and the labelling alignment.

8.3.2 "Check" menu item

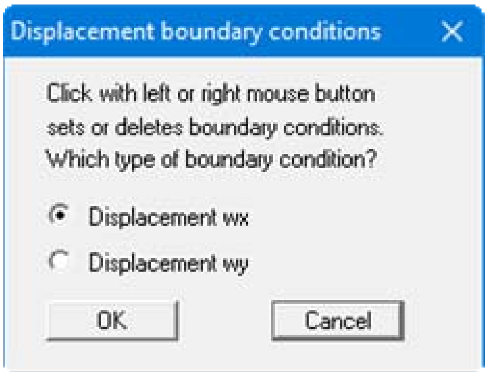
With this menu item, you can get an overview on the position of all boundary conditions.



The boundary conditions selected in the dialog box are drawn using the declared letter at the nodes for which you have entered boundary conditions.

8.3.3 "Individual displacement BC" menu item

After calling up this menu item a displacement boundary condition in x or y direction can be set with the left mouse button, or be deleted with the right mouse button.



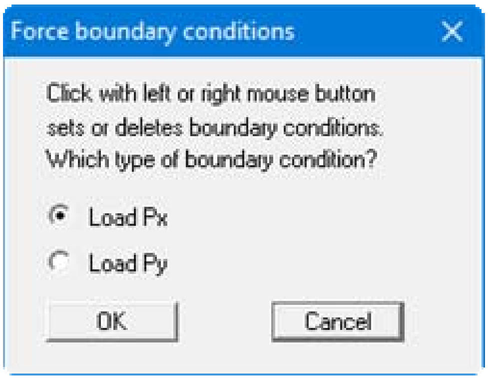
8.3.4 "(Displacement BC) In section" menu item

After selecting this menu item all nodes in a section can be assigned a displacement boundary condition, or have one cancelled. Four points must be given by clicking with the left mouse button in an anti-clockwise direction. The right mouse button undoes the selected point. The definition is with reference to all nodes within the quadrilateral.

8.3.5 "Point loads" menu item

The dimension of point loads is e.g. kN/m.

With this menu item the point loads in the FEM system are agreed upon, in analogy to the displacement boundary conditions.

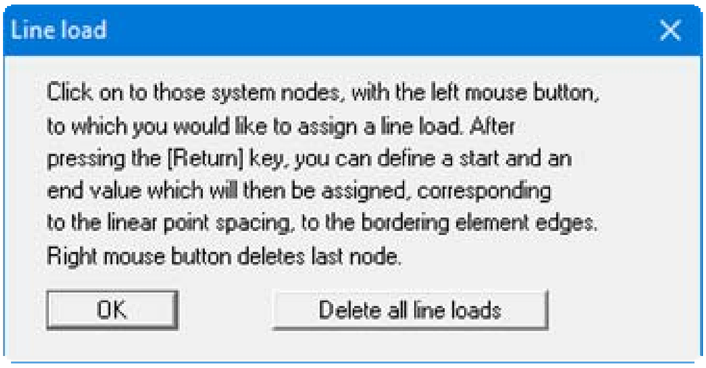


The case of a freely supported boundary is automatically considered by the finite-element-method. Valid is, that all system boundaries or partial system boundaries, which have no displacement or force boundary conditions, are automatically freely supported.

8.3.6 "Line loads" menu item

The dimension of line loads is e.g. kN/m/m = kN/m².

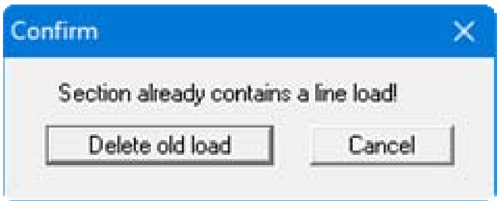
General information can be found in the menu item "**Point loads**" (see Section 8.3.5). The procedure for defining line loads is explained in the dialog box.



After clicking system nodes and confirming the selection with the [**Return**] key, a dialog box opens for defining the start and the end value. The intermediate boundaries will be assigned the appropriate value in a linear relationship to the point spacing.

If line loads have already been defined an additional "**Delete all line loads**" button appears in the info box after selecting this menu item again. If you exit the dialog box by using this button, all line loads are deleted and a new line load can be defined.

If you need to delete individual line loads or sections of them, click the system nodes again, accept with the [**Return**] key and exit the start and end value dialog box by pressing "**OK**". The following dialog box now appears:

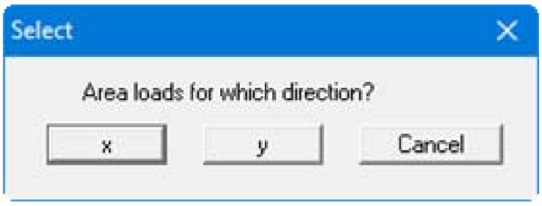


Click the "**Delete old load**" button until all sections of the marked line load are deleted and then define the new line load if required. If you exit the box by pressing "**Cancel**" before you are finished, the undeleted boundaries will remain in place.

8.3.7 "Area loads" menu item

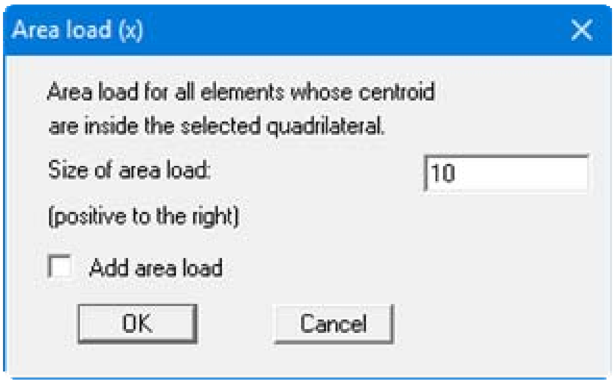
The dimension of area loads is e.g. kN/m²/m = kN/m³.

You can assign certain elements area loads in x and y direction.



Area loads act in the representation plane. Own-weight loads are e.g. area loads in y-direction. The own-weight of triangular elements can, however, be defined more easily via the definition of "γ" in the menu item "**System/Material properties**" (see Section 8.4.3).

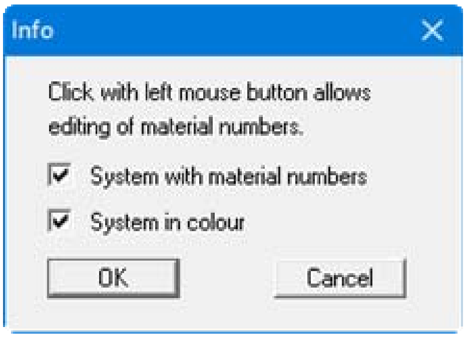
Area loads in x-direction are rather rare in soil mechanics. An area load in x-direction would be e.g. a centrifugal force.



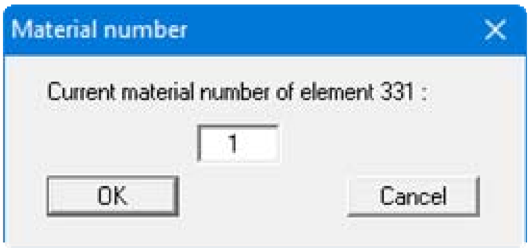
If you select the "**Add area load**" check box, the value entered will be added to any area loads already present.

8.3.8 "Individual materials" menu item

You can modify the materials of individual triangular elements. Before you can select the required elements with a mouse click, you will first be asked how the FEM system should be displayed on the screen.



After specifying the display settings and clicking on the required triangular element, you can assign a new material number in the following dialog box.



For each material number you enter, the program assigns a data row to enter properties (Young's modulus, ν, γ, etc.) in the menu item "**System/Material properties**" (Section 8.4.3), which can be different for each material.

8.3.9 "(Material) In section" menu item

The menu item corresponds to the previous one. However, you can edit the material numbers of several FEM elements at the same time by defining a section window. All triangular elements whose centroids are within the section are modified.

8.3.10 "Beams" menu item

After you have defined a FEM mesh, you can define beams along nodes of the FEM mesh which can later be considered when analysing stiffness.

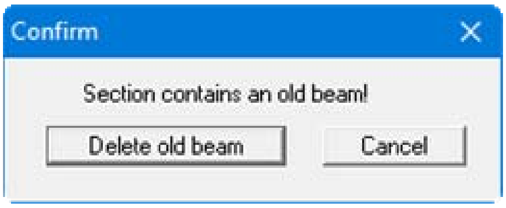
This is useful e.g. when taking the influence of piles or similar objects into account.

To enter the beams, left-click on system nodes and define beam elements. You then assign these beams a beam material number.

If, e.g., you have several piles in the system, with differing thicknesses and/or widths, you can assign a different material number for each type. Depending on the number of beam materials, the same number of lines appears in the menu item "**System/Material (beams)**" for definition of beam materials (see Section 8.4.4).

The defined beams are shown in the graphics with the appropriate material number. The colour and line strength can be edited in the menu item "**Graphics preferences/Pen colour and width**" (section 8.5.3).

If you would like to delete specific beams again, click the beams a second time. Enter any beam material number. You will then see the following dialog box:



If you press the "**Delete old beam**" button, you can then enter a new beam element by selecting the menu item once again.

8.3.11 "Delete all" menu item

You can delete all defined beams.

8.4.1 "Info" menu item

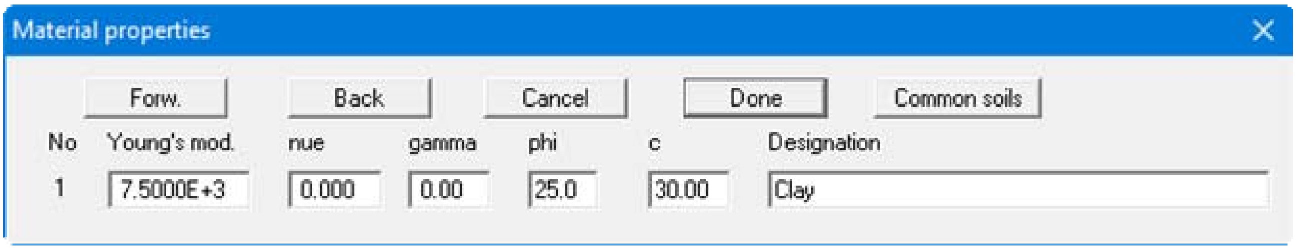
After selecting this menu item information will be shown on the selected system, with the number of triangular elements, nodes and boundary conditions.

8.4.2 "Project identification" menu item

You can enter a description of the current system. This text appears in the ***General legend*** (see Section 8.5.8).

8.4.3 "Material properties" menu item

You can edit the characteristics. A dialog box appears with a number of input lines dependent on the defined number of materials.

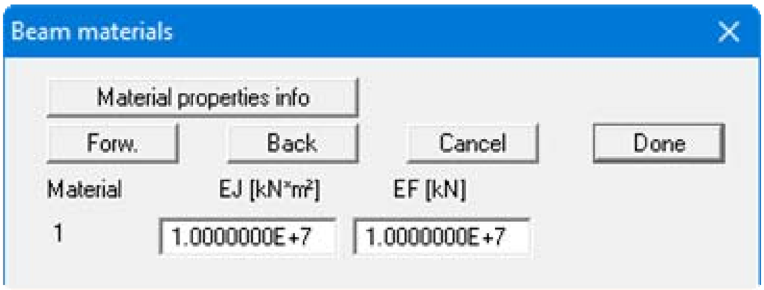


Numerical input for the friction angle phi and the cohesion c are important for analysis of the flow condition after Mohr/Coulomb.

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

8.4.4 "Material (beams)" menu item

You can edit the characteristics of any beams defined. A dialog box appears with a number of input lines dependent on the number of beam materials selected.



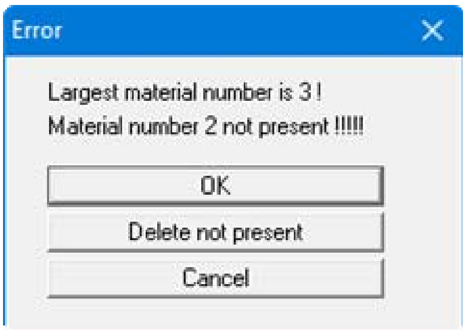
The following designations are valid:

- EJ = beam bending resistance
- EF = beam elasticity coefficient
- E = Young's modulus of beam
- J = moment of inertia of beam
- F = beam area

You can display a legend containing the material properties of any existing beams (EJ = beam bending stiffness, EF = beam tensile stiffness, see Section 8.5.10).

8.4.5 "Test" menu item

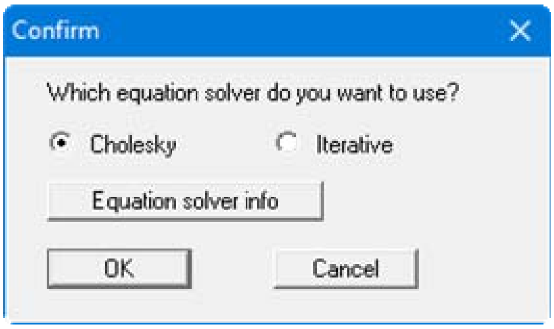
You can then have various system conditions tested prior to analysis. Here, e.g. any irregularities in soil numbering can also be displayed, as the following error box illustrates:



For the purpose of further testing, you can select between "**Node spacing**" and/or "**Overlapping**" of the defined triangular elements of the specified system. The test can be aborted at any time by right-clicking.

8.4.6 "Analyse" menu item

Two equation solvers are available for system analysis:



- **"Cholesky"**
  The Cholesky equation solver is characterised by its good robustness.
- **"Iterative"**
  The iterative equation solver is characterised by its extremely low memory requirements.

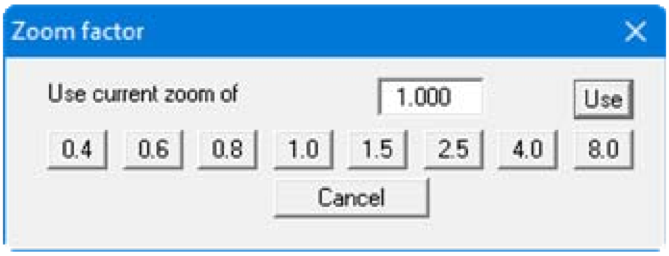
After selecting the equation solver and starting you are continuously informed about the state of the analysis in an info box. After completing the analysis, you will see information on the analysis time and can then evaluate the results.

8.5 Graphics preferences menu

8.5.1 "Refresh and zoom" menu item

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

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



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

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

8.5.2 "Zoom info" menu item

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

8.5.3 "Pen colour and width" menu item

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

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

8.5.4 "Legend font selection" menu item

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

8.5.5 "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:

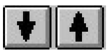
- Objects created with **Mini-CAD** are based on the coordinate system (generally in metres), in which the drawing is produced, and are shown accordingly. You should use the "**Mini-CAD toolbar**" when you wish to add information to the system (for example, labelling of slope inclinations or the location 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.

8.5.6 "Toolbar preferences" menu item

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

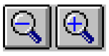
At the bottom of the program window you find a status bar with further information. You can also activate or switch off the status bar here. The preferences will be saved in the "**GGU-ELASTIC.alg**" file (see menu item "**Graphics preferences/Save graphics preferences**") and will be active at the next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

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

 "**Next page**"/"**Previous page**"
Using this icon, you can navigate between the individual pages in the ***tabular representation***.

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

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

 **"Zoom (-)"/"Zoom (+)"**
With the zoom functions you can zoom in or out of parts of the image, by clicking the left mouse button.

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

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

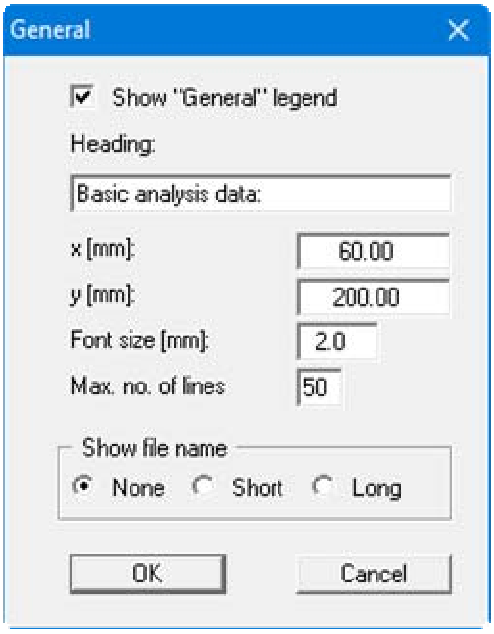
8.5.7 "3D toolbar" menu item

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.

8.5.8 "General legend" menu item

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



In addition to the heading, this legend contains information on the current graphics and preferences, e.g. which state variable is currently displayed as contours. For example, if you display the velocity field you will see information on the velocities scale.

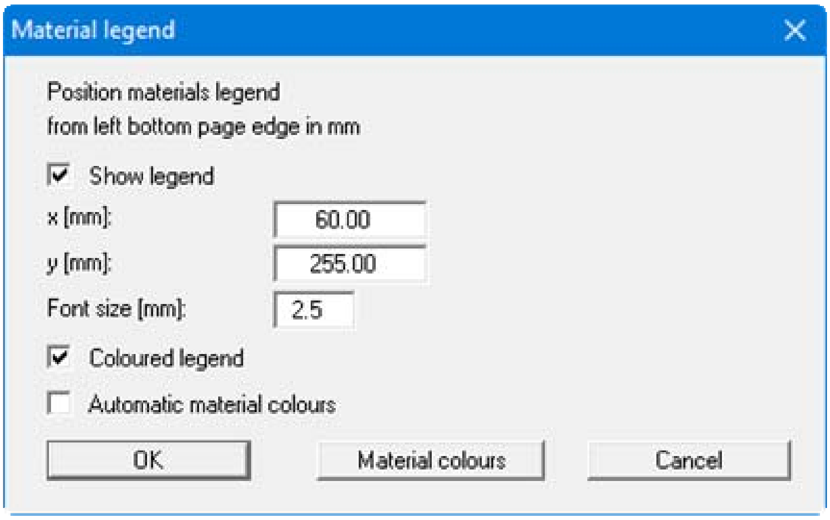
You can define and edit the position of the legend using the values "**x**" and "**y**". You control the size of the legend using "**Font size**" and "**Max. no. of lines**"; where necessary, several columns are used.

> 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 the file name. Any project identification entered (see Section 8.1.1) will also be shown in the general legend.

8.5.9 "Material legend" menu item

A legend with the material properties will be displayed on your output sheet if you have clicked the "**Show legend**" check box. Using this menu item you can alter the type of presentation or turn off the legend completely.



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 while holding the left mouse button.

- "**Coloured legend** "
  The material colours are displayed in the legend. Otherwise, they will be numbered.
- "**Automatic material colours**"
  The materials/soils are assigned material colours automatically by the program. If the check box is not selected, the material colours individually defined using the "**Material colours**" button will be adopted.
- "**Material colours**"
  You will see a dialog box, in which you can define your preferences. After clicking the button with the desired number you can assign each material/soil a new number or reorganise using the "**Material colours/Reorganise**" command button. You can save your colour preferences to a file with "**Material colours/Save**" and use them for different systems by means of the "**Material colours/Load**" command button. In the lower group box you can also transfer the colour preferences to the Windows colour management dialog box, or vice versa, as user-defined colour preferences for example. You can read a further description by pressing the "**Info**" button.

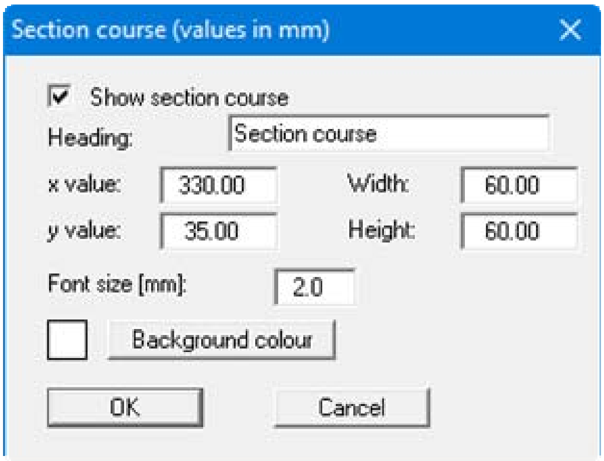
8.5.10 "Beam legend" menu item

You can display a legend containing the material properties of any existing beams (EJ = beam bending stiffness, EF = beam tensile stiffness, see Section 8.4.4).

Even when the "**Show legend**" check box is activated, visualisation only occurs if beams have actually been defined.

8.5.11 "Section course legend" menu item

If you have selected section functions when evaluating the analysis results, e.g. values in the node section (see Section 8.7.7), you can use this menu item to display and edit a legend containing the section course



The position of the legend can be defined and edited using the values "**x**" and "**y**", "**Width**" and "**Height**". The legend heading and font size can be modified as well as a background colour.

Even when the "**Show section course**" check box is activated, visualisation only occurs if you perform a section evaluation.

8.5.12 "Move objects" menu item

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

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

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

8.5.14 "Load graphics preferences" menu item

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

8.6 *Page size + margins menu*

8.6.1 "Auto-resize" menu item

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

8.6.2 "Manual resize (mouse)" menu item

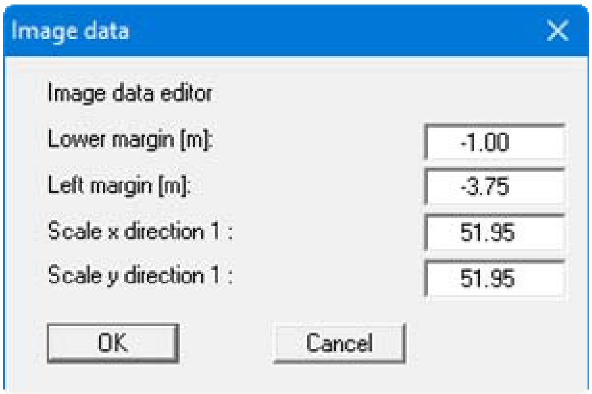
You can use the coordinates of a section of the visualisation as the new image coordinates by marking the desired area with the mouse, pressing the left mouse button and holding the [**Ctrl**] and [**Shift**] keys. The scales of the x- and y-axes are adjusted accordingly. If the previous proportions (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.

8.6.3 "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 8.6.4).

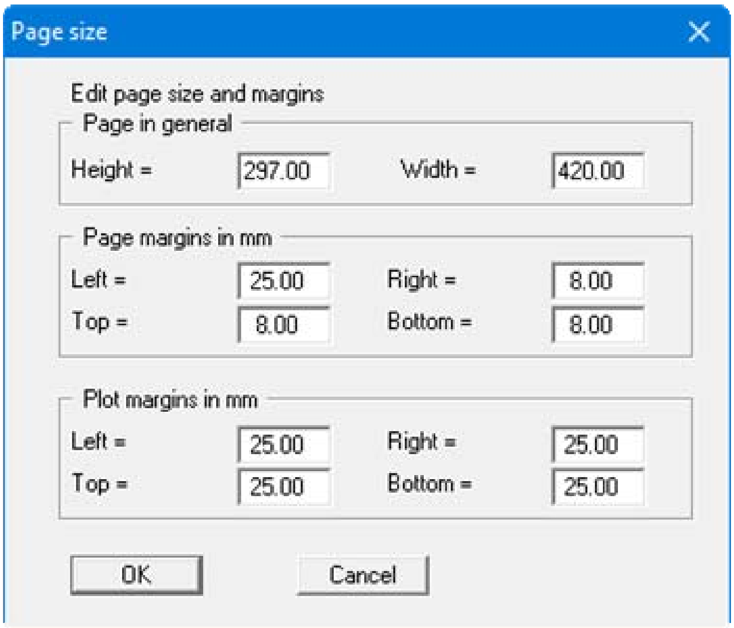
When displaying the FEM mesh, you will see the following dialog box, for example, in which you can alter the size and orientation of the visualisation by entering the appropriate numbers.



The dialog box is always adapted in line with the evaluation that is currently displayed on the screen. For example, if you are visualising "**Values in node section**", you will see an adapted dialog box.

8.6.4 "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 using the "**With borders**" check box in the "**Page size + margins/Margins and borders**" menu item (see Section 8.6.6).
- "**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 deactivating the "**With margins**" check box in the "**Page size + margins/Margins and borders**" menu item (see Section 8.6.6).
- The "**Plot margin**" define a set distance between the page margin and the actual ***drawing area*** in which the graphical evaluation of your input is presented.

8.6.5 "Font size selection" menu item

You can edit font sizes for labelling the various graphical elements. The font sizes of text within legends are edited in the respective legend editor. Just double-click in a legend to do this.

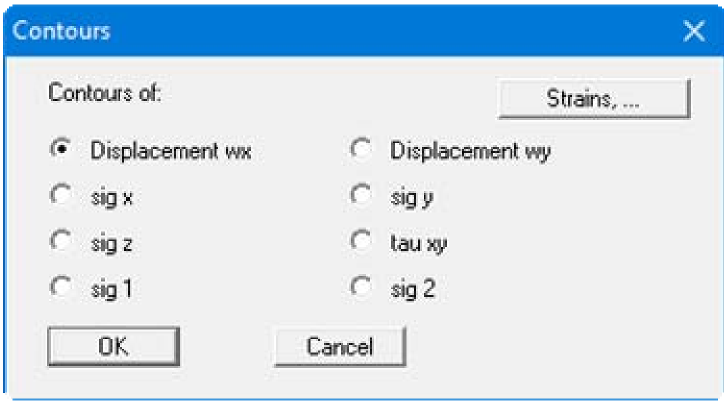
8.6.6 "Margins and borders" menu item

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

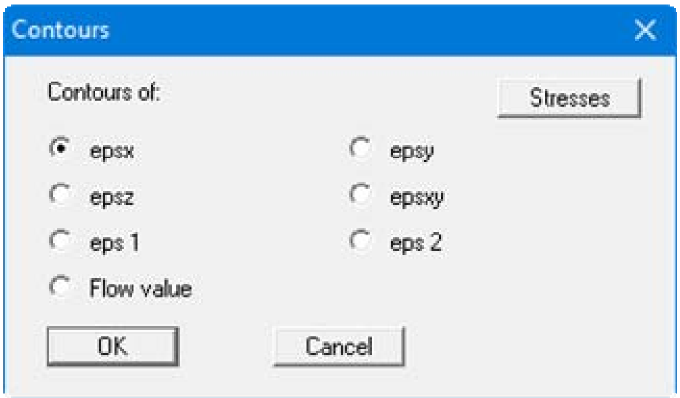
8.7 Evaluation menu

8.7.1 General

Numerous options are available for evaluation and graphical representation of the analyses. If you select a contour, circle chart or section visualisation, the respective dialog box opens, in which you first select the state variable to be visualised.



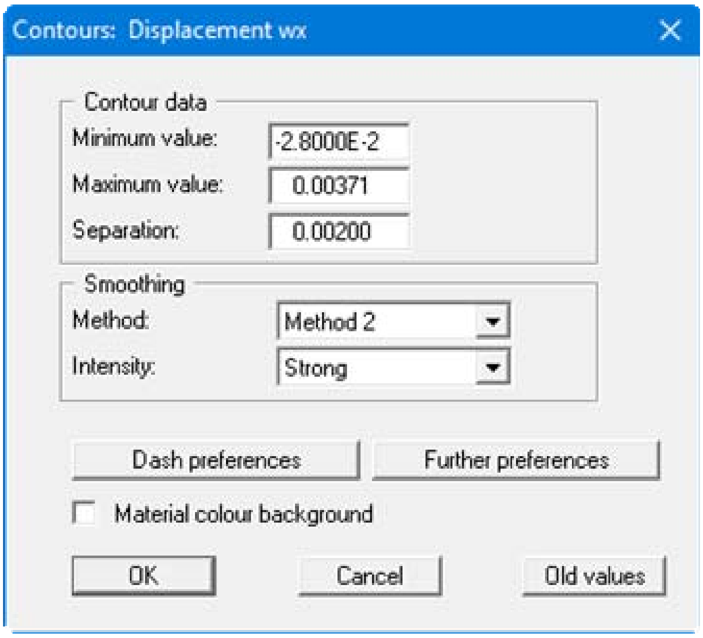
If you would like to have strains displayed, click on the "**Strains,..**" button and you will see:



If you select the "**Stresses**" button, you will once again see the dialog box for selection of stresses and displacements (see above).

8.7.2 "Normal contours" menu item

Using this menu item it is possible to visualise the lines of equal state variables.



In the example dialog box for displacement wy, the program shows you the existing smallest and largest values and the spacing of the contour lines. If you want the visualisation to begin at a different value, the initial value can be entered here. You can also vary the spacing, for example, in order to reduce the number of contour lines drawn.

When you select this menu item 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. This information is saved with the record.

For the smoothing out method, you can choose between "**Do not smooth**", "**Method 1**" and "**Method 2**", and then define a smoothing intensity to be used for each method.

There is no optimum method. The method suitability depends on the type of values.

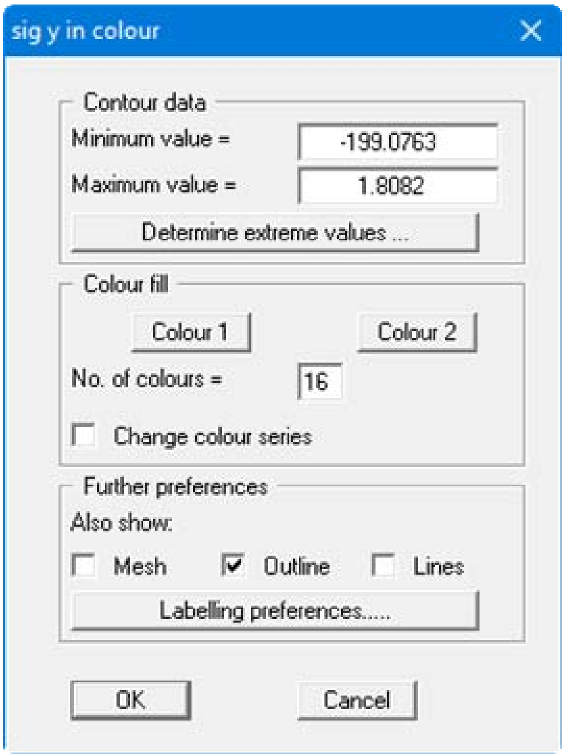
Using the "**Dash preferences**" button, you can achieve a clearer visualisation of a large number of contour lines, for example by defining different dashes for selected lines. The following dialog box shows an example in which a continuous contour line and a dashed contour line are alternately drawn.

The "**Further preferences**" button opens a dialog box in which you can additionally define the contour visualisation, among other things, the type of labelling.

Using the "**Material colour background**" button, you simultaneously activate the system colour display.

8.7.3 "Coloured contours" menu item

With this menu item you can have lines of equal state variables displayed in colour.



- "**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.

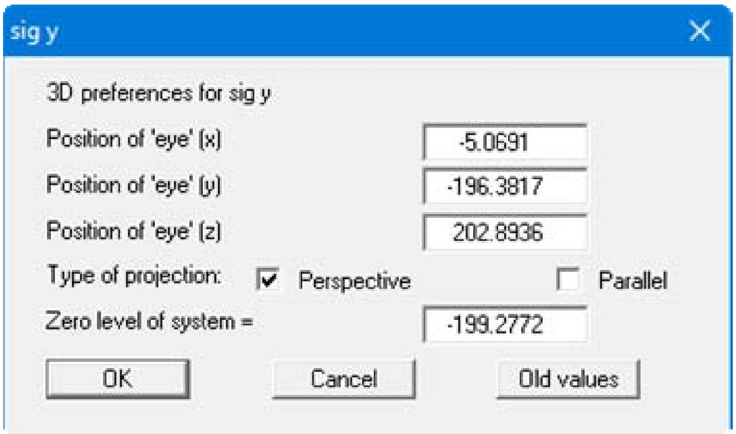
- "**Colour fill**" group box
  You can control the colour subdivisions of the contour diagram using "**No. of colours**". In the example above, 16 colours will be displayed between "**Colour 1**" and "**Colour 2**". The default setting is a colour course from red to blue. These colours can be edited 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.

The colours will be drawn after confirmation with "**OK**". A colour bar at the right edge of the sheet allows correlation between the colour and the corresponding value. 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 8.6.4).

8.7.4 "3D contours" menu item

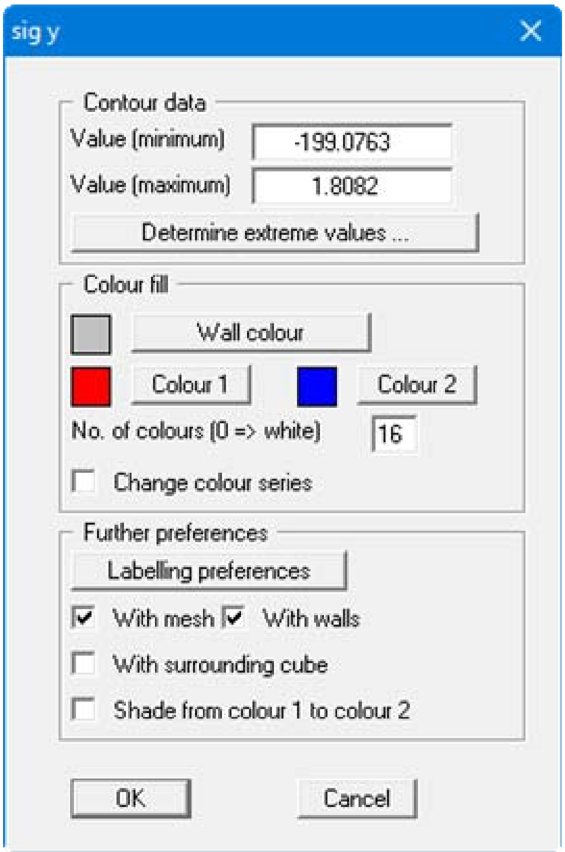
With this menu item you can have lines of equal state variables for the current nodes displayed three-dimensionally. In the following dialog box you can select from perspective and parallel projection, as well as selecting the position of the "eye". The values are displayed above and below the base level (z ordinate) in three dimensions.



In general, the program will make a sensible suggestion for the input values in the dialog box, so you will not normally need to make alterations.

Whenever you select the menu item "**Evaluation/3D contours**", the values determined automatically by the program are displayed first. Using the "**Old values**" button, the preferences entered for the previous 3D visualisation are adopted. This information is saved with the record.

After leaving the dialog box by pressing "**OK**" you will see the following dialog box. It greatly resembles the box shown in the menu item "**Evaluation/Coloured**" (see Section 8.7.3). Below, therefore, only the buttons and check boxes not previously described will be explained.

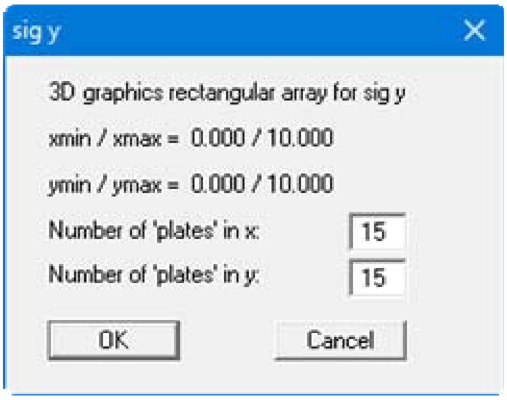


- "**Wall colour**"
  When the corresponding check box is activated, you can select a colour for the walls that surround the three-dimensional visualisation.
- "**With walls**"
  You can activate enclosure of the three-dimensional visualisation by walls.
- "**With surrounding cube**"
  You can also specify whether an enclosing cube should be displayed; in some cases this improves the 3D effect.
- "**Shade from colour 1 to colour 2**"
  The 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.

If the triangular mesh has a very irregular plan, the 3D impression will probably not be optimal. It is then probably better to use the menu item "**Evaluation/3D array contours**".

8.7.5 "3D array contours" menu item

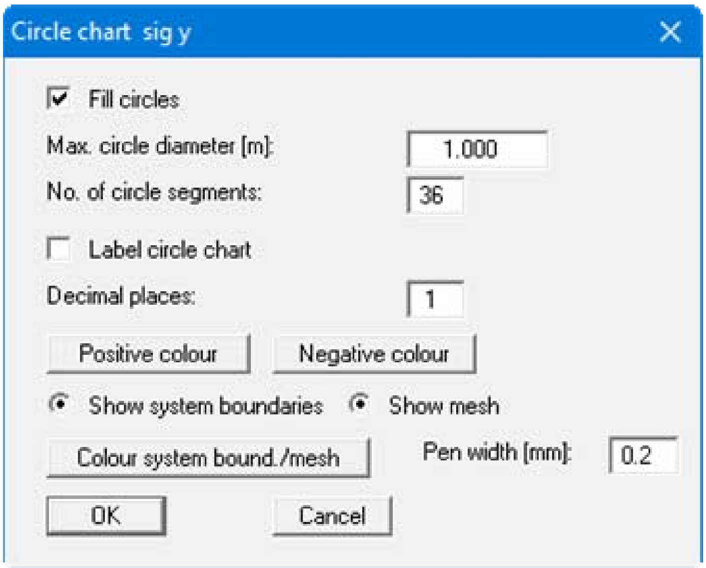
The 3D effect is usually lost in an irregular triangular mesh. The "**Evaluation/3D array**" menu item is useful here. After selecting the required parameter the following dialog box opens where you can define the array.



After leaving the dialog box by pressing "**OK**" you will at first see the same box as that described in "**Evaluation/3D contours**". The program calculates the state variables at the array points by linear interpolation of the results of the FEM analysis. You then see the dialog box for coloured contours preferences as described in "**Evaluation/3D contours**" (see Section 8.7.4) and so on.

8.7.6 "Circles" menu item

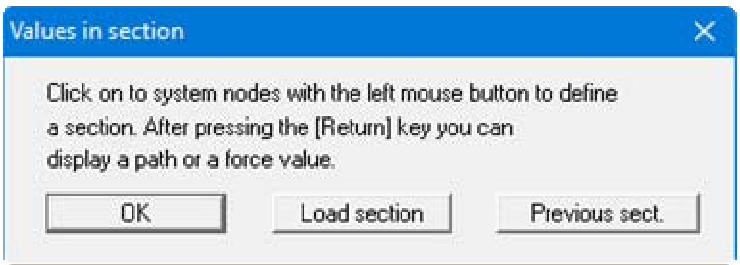
After selecting the required parameter, circle chart can be generated. The selected values are displayed as circles of varying (value-dependent) circles.



If you activate the check box the circles with positive vales will be filled with the "**Positive colour**" and those with negative values with the "**Negative colour**". With "**Max. circle diameter**" you set the circle diameter for the maximum state variable. With "**No. of circle segments**" you set a value for the circle resolution. For a value e.g. of 3, triangles will be drawn. You can label each circle with state variable value. With the "**Positive colour**" and "**Negative colour**" buttons, you can edit the colour fill for the circles. In addition to the circles, the system boundaries or the FEM mesh can be shown. With "**Colour system bound./mesh**" and pen width you can set preferences for graphical presentation.

8.7.7 "Values in node section" menu item

You can display state variables in a section. After going to this menu item, you will see the following explanation of how to define a section.

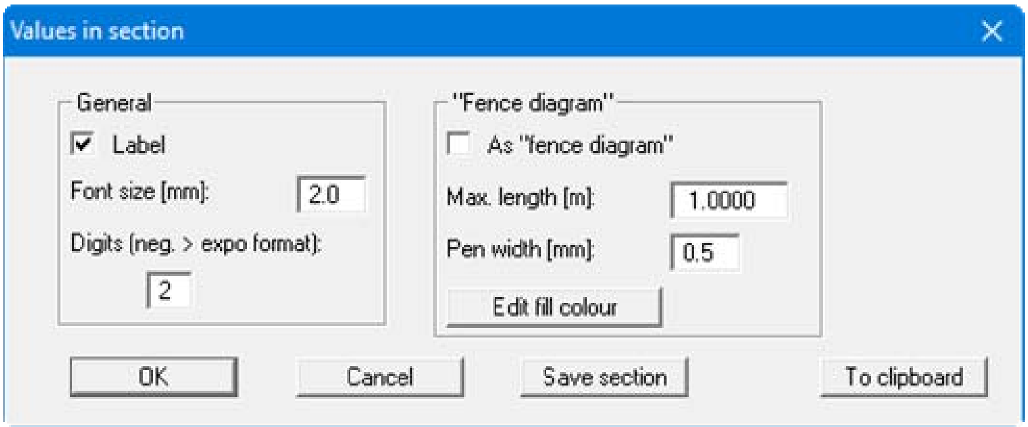


If you click "**OK**", you can define a new section course. The FEM mesh is displayed on the screen to assist you. You have the option of selecting each node for representation in section individually. However, you can also select the starting point and the end point. In this case, the system nodes lying along the shortest route from the starting point to the end point will be automatically included in the section course. The last point defined can be reset by right-clicking.

After defining the node sequence or importing a previously defined node sequence, the section is always confirmed by pressing the [**Return**] or the [**Enter**] key.

It is possible, by clicking "**Load section**", to load a section course saved during a previous session. If you have defined a section during the current session, the additional button "**Prev. section**" is shown. The respective previous section can then be opened. Both actions will show you a visualisation of the section line in the FEM mesh and you then only need to confirm this using the [**Return**] key.

If you have imported or specified the section course, select the state variable to be displayed in the familiar dialog box and then the following dialog box opens, in which you can enter settings for sectionas fence diagram visualisation.



Define the font size and the number format for section labelling in the "**General**" group box. Your system can also be visualised as a fence diagram by activating the "**As fence diagram**" check box. Here, you will also find additional fence diagram settings.

Using the "**Save section**" button, you can save your section to a file so that the exact section is available again via the "**Load section**" button in the dialog box at a later date.

Using the "**Clipboard**" button, you can paste the data for nodes, x, y, wx, wy, sigx, sigy, sigz, tauxy, epsx, epsy, epsz and epsxy into another document via the Windows clipboard.

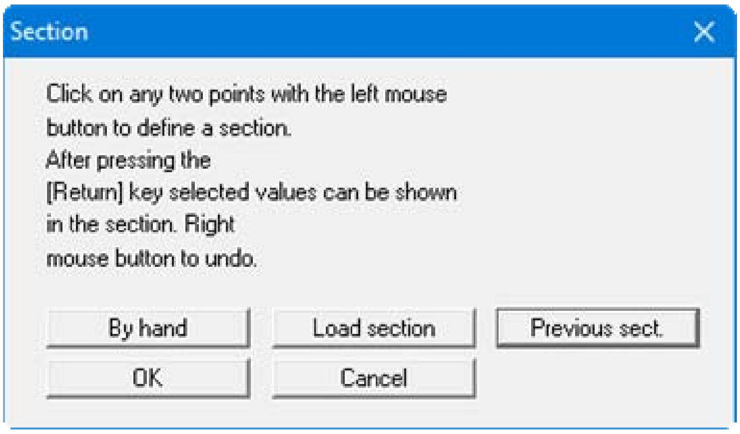
8.7.8 "Position (of node section)" menu item

You can have the position of the section selected using the previous menu item displayed and, if required, printed.

You can see the same section visualisation in a smaller format in the section legend, which you are presented together with the "**Values in node section**" visualisation. The section legend must be activated to achieve this (see Section 8.5.11).

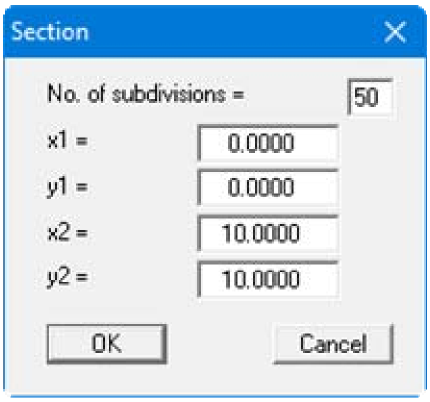
8.7.9 "Any section" menu item

After selecting the required state variable, you will see a dialog box with an explanation of how to define the section course.



With this evaluation method, the section course is no longer tied to nodes, but consists of a start and end point.

With the "**By hand**" button you can enter the start and end points as numerical values directly, without having to use the mouse.

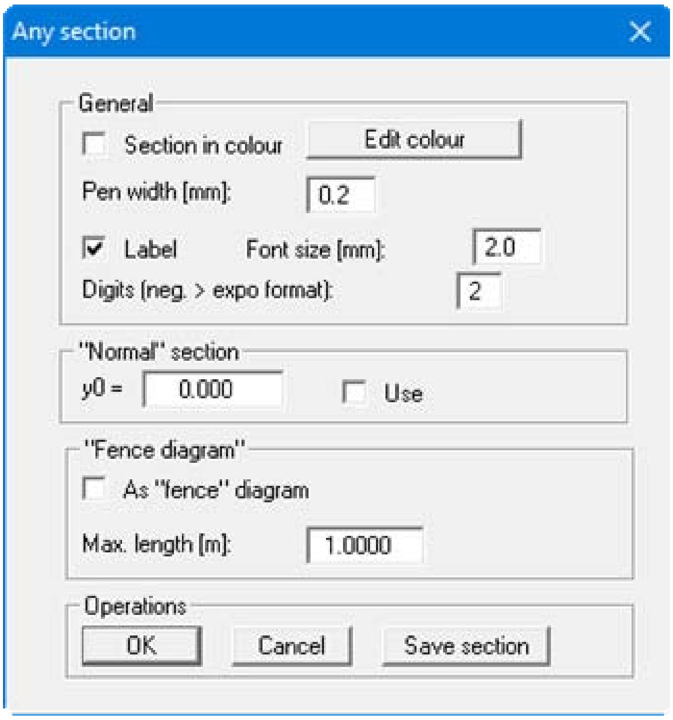


The no. of subdivisions defines at how many points section values are to be calculated.

If you want to define the section graphically, click "**OK**". You can click the start and end points at any location using the mouse. You can enter the number of subdivisions in the subsequent dialog box.

If you have previously saved a section you can reload it using "**Load section**". The "**Previous sect.**" button is only available if a section has already been defined.

After leaving the dialog box for defining the section by hand or defining the subdivisions for the other methods by pressing "**OK**", a dialog box opens in which you can click and enter settings for section visualisation.



You can now specify whether the section should be displayed in colour. If you do not require a colour-filled section, you can display a thicker section line, for example, using the "**Pen width**" input box. You can adapt the colour settings for the colour fill or section line to suit your needs using the "**Edit colour**" button.

You can also define the intercept of the x-axis with the y-axis for a ***normal*** section visualisation by specifying "**y0 =**". In this case the "**Use**" check box must be activated. If you do not use this option, the x axis will intercept the y axis at the smallest occurring y value.

Your system can also be visualised as a fence diagramas fence diagram by activating the "**As fence diagram**" check box. In the corresponding input box, enter the maximum length to be displayed.

Using the "**Save section**" button, you can save your section to a file so that the exact section is available again via the "**Load section**" button in the dialog box at a later date.

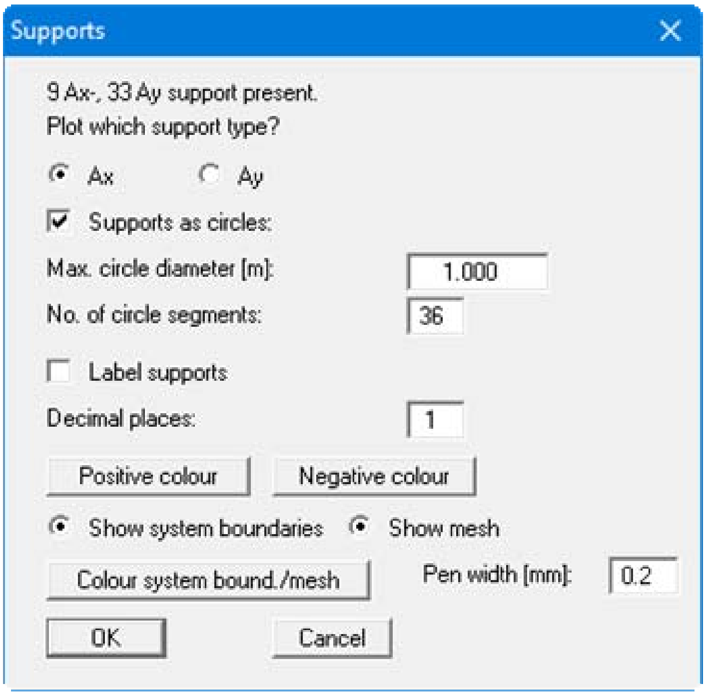
8.7.10 "Position (of any section)" menu item

You can have the position of the section selected using the previous menu item displayed and, if required, printed.

You can see the same section visualisation in a smaller format in the section legend, which you are presented together with the "**Values in node section**" visualisation. The section legend must be activated to achieve this (see Section 8.5.11).

8.7.11 "Support" menu item

You can generate a circle chart of the bearing forces using this menu item.



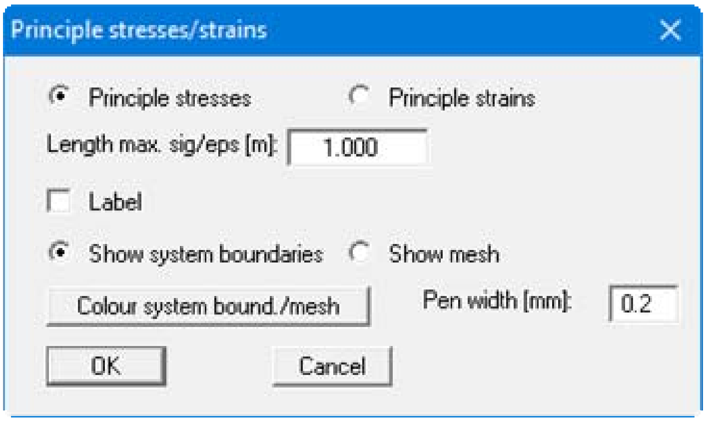
In the upper part of the dialog box you set the type of support to be determined.

- Ax = support due to displacement boundary condition wx
- Ay = support due to displacement boundary condition wy

All further input is identical to input for the menu item "**Circles**" (see Section 8.7.6).

8.7.12 "Principle stresses/strains" menu item

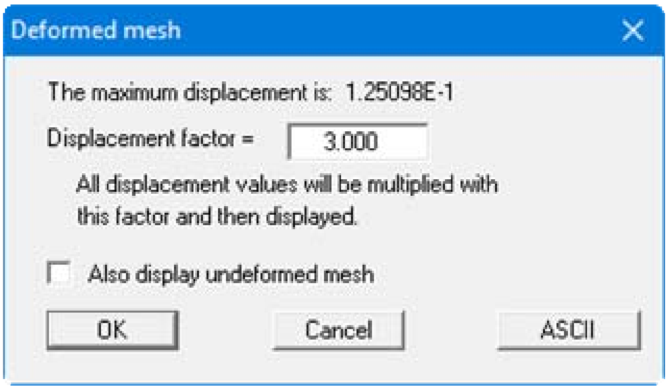
The principal stresses and principal strains can be visualised as value-dependent dashes. Select the appropriate check box in the subsequent dialog box to achieve this.



Define the maximum dash length in the following input box as the quotient "**sig/eps [m]**". The line colour and line thickness can be specified separately for positive and negative principal moments in the menu item "**Graphics preferences/Pen colour and width**" (Section 8.5.3). The remaining settings are already known from the explanations in the previous section.

8.7.13 "Deformed mesh" menu item

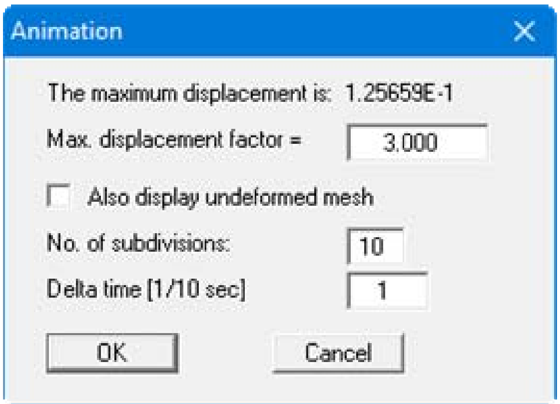
You can have the deformed mesh displayed. As deformations are generally small in comparison to the system dimensions, a factor can be given, with which the currently calculated displacements will be multiplied.



To add clarity to the deformations, the undeformed mesh can also be displayed. The line colour and line thickness can be specified separately for the deformed mesh in the menu item "**Graphics preferences/Pen colour and width**" (Section 8.5.3).

8.7.14 "Animation" menu item

The system deformations can be represented in sub-steps.

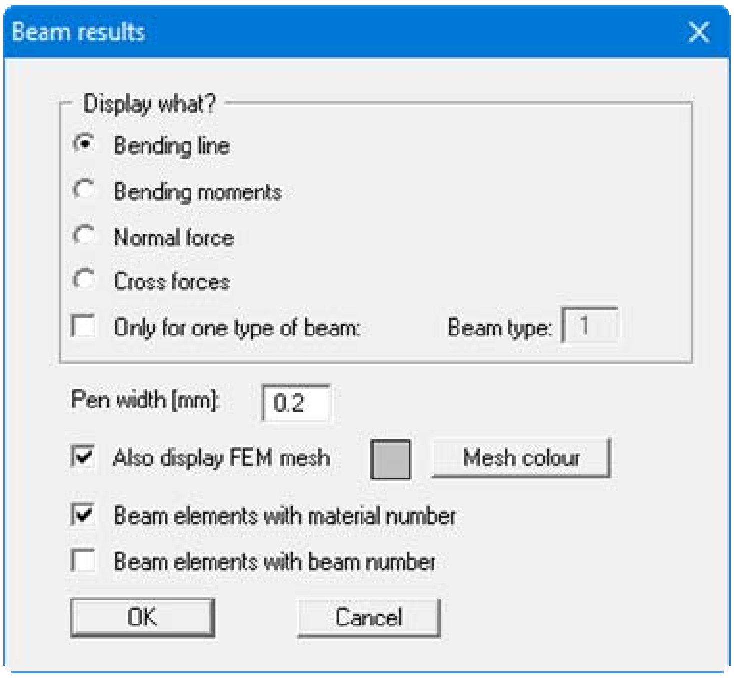


Similar to the previous menu item, the system deformations can be multiplied by a factor and displayed together with the undeformed mesh for emphasis.

However, not only the final state is visualised, but in "**Number of subdivisions**" and "**Delta time [1/10 sec]**" you also specify the substeps with which to display the deformation. This creates an animation-like effect. A running animation can be cancelled by clicking the left mouse button.

8.7.15 "Beams" menu item

If beams are present in the system, you can have their state variables graphically presented. Select the required state variable in the following dialog box:

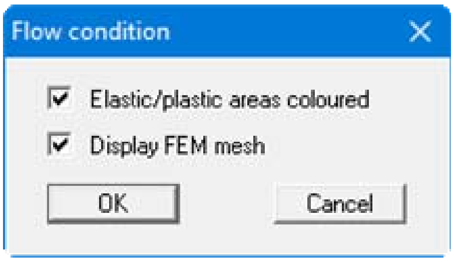


In the lower group box, you can define the familiar visualisation settings, with the addition here of the check box for visualisation of the "**Beam elements with material number**".

After confirming with "**OK**", you will see further dialog boxes for all visualisations, in which you can define the length of the maximum state variable and the line labelling.

8.7.16 "Flow" menu item

From the calculated stresses and the shear parameters φ and c, the flow condition after Mohr/Coulomb is calculated and displayed.



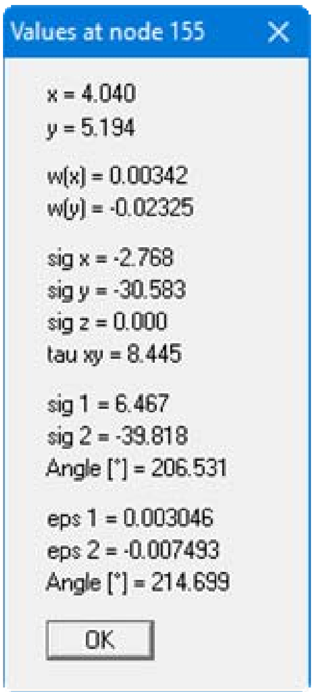
The colours with which elastic and plastic areas are represented can be edited in the menu item "**Pen colour and width**" with the "**elastic**" and "**plastic**" buttons. (see Section 8.5.3)

8.7.17 "Mohr" menu item

After selecting this menu item, the associated shear plane and Mohr's stress circle can be visualised by clicking on any element of the FEM mesh.

8.7.18 "Individual values" menu item

By clicking a node with the left mouse button, you can view all data for this node, e.g. here for node no. 155.



8.8 Info menu

8.8.1 "Copyright" menu item

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

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

8.8.2 "Maxima" menu item

In a box you will see the default program maxima for the nodes and elements of the FEM mesh.

8.8.3 "Help" menu item

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

8.8.4 "GGU on the web" menu item

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

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

8.8.5 "GGU support" menu item

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

8.8.6 "What's new?" menu item

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

8.8.7 "Language preferences" menu item

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

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

9 Tips and tricks

9.1 Keyboard and mouse

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

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

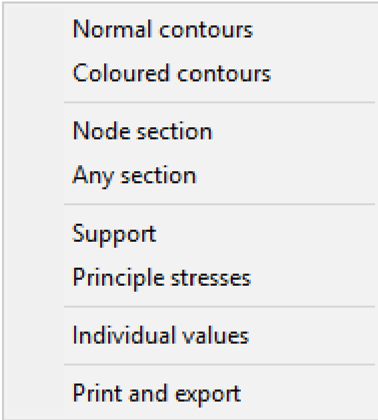
Change system graphics (new values can be checked in "Page size + margins/Manual resize (editor)"):

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

Change screen coordinates:

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

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



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

You can simplify system input by integrating graphics or DXF files into the system via the **Mini-CAD** module (see Mini-CAD manual). The size of the graphics or the DXF data can be imported to the correct scale. However, this is not absolutely necessary for mesh generation. If you do not model the system to scale using a graphics file, you can perform a scale correction after input is complete using the menu item "**FEM mesh/Change**", "**Via equation**" button (see Section 8.2.5).

9.2 Function keys

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

- [**Esc**] refreshes the screen contents and sets the screen back to the given format. This is useful if, for example, you have used the zoom function to display parts of the screen and would like to quickly return to a complete overview.
- [**F1**] opens the manual file.
- [**F2**] refreshes the screen without altering the current magnification.
- [**F3**] opens the menu item "**FEM mesh/Define nodes**".
- [**F4**] opens the menu item "**FEM mesh/Manual mesh**".
- [**F5**] opens the menu item "**System/Analyse**".
- [**F6**] opens the menu item "**FEM mesh/(Refine) Section**".
- [**F7**] opens the menu item "**FEM mesh/(Refine) All**".
- [**F9**] activates the menu item "**Page size + margins/Auto-resize**".
- [**F11**] activates the menu item "**Graphics preferences/Move objects**".

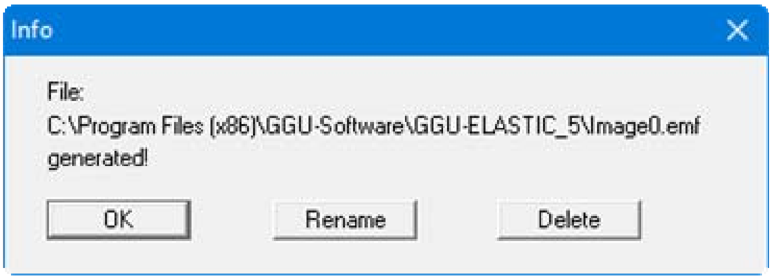
9.3 "Copy/print area" icon

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

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

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

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



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

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

10 Index

G

H

I

L

M

N

O

P

Q

R

S

T

U

V

W

Y

Z