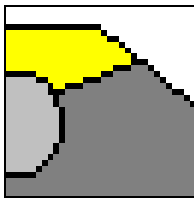


Calculation of plane and axis-symmetrical strain
conditions (FEM)

GGU-ELASTIC

VERSION 1



Last revision: January 1997
Copyright and Author: Prof. Dr.-Ing. Johann Buss, Braunschweig

Contents

1	Preface	5
2	Installation	5
3	Program concept	5
4	Short description	6
5	Theoretical principles	8
5.1	General	8
5.2	Signs and designations.....	10
6	Menu item File	11
6.1	Menu option „New”.....	11
6.2	Menu option „Load file”.....	11
6.3	Menu option „Save file”.....	11
6.4	Menu option „Read ASCII file”.....	12
6.5	Menu option „Save ASCII file”.....	12
6.6	Menu option „Protocol output“	13
6.7	Menu option „Printer preferences”.....	14
6.8	Menu option „Start”.....	14
7	Menu item FE grid	16
7.1	Menu option „Preferences“ and „FE grid“	16
7.2	Menu option „Outline”.....	16
7.3	Menu option „Set nodes“	16
7.4	Menu option „Edit nodes“.....	17
7.5	Menu option „Move nodes“	18
7.6	Menu option „Edit nodes“.....	18
7.7	Menu option „Raster“	18
7.7.1	„Regular“	19
7.7.2	„Irregular“	20
7.8	Menu option „By hand“	20
7.9	Menu option „Automatic“	20
7.10	Menu option „Round off“	21
7.11	Menu option „Delete“.....	21
7.12	Menu option „Optimise“.....	21
7.12.1	Optimise diagonals”	22
7.12.2	Optimise topology	22
7.13	Menu option „Refine“	22
7.13.1	Method 1	24
7.13.2	Method 2	25
7.13.3	Method 3	25
8	Menu item Boundary	26
8.1	Menu option „Preferences”	26
8.2	Menu option „Check”.....	26
8.3	Menu option „Individual path”	26
8.4	Menu option „In section”	26
8.5	Menu option „Single loads“	27

8.6	Menu option „Line loads“	27
8.7	Menu option „Area loads“	28
8.8	Menu options „Individual material“ and „In section“	29
8.9	Menu option „Rods“	29
8.10	Menu option „Delete all“	30
9	Menu item System	30
9.1	Menu option „Info“	30
9.2	Menu option „Data set designation“	30
9.3	Menu option „Material characteristics“	30
9.4	Menu option „Material (rods)“	31
9.5	Menu option „Test system“	31
9.6	Menu option „Calculate“	31
10	Menu item View	32
10.1	Menu option „Refresh“	32
10.2	Menu option „Zoom“	32
10.3	Menu option „Pens“	33
10.4	Menu option „Font“	33
10.5	Menu option „Mini-CAD and CAD for header data“	33
10.6	Menu option „Smarticons“	34
10.7	Menu option „3D Smarticons“	34
10.8	Menu option „General legend“	35
10.9	Menu option „Materials legend“	35
10.10	Menu option „Rod legend“	36
10.11	Menu option „Move legends“	36
10.12	Menu options „Save and Load preferences“	36
11	Menu item Page	36
11.1	Menu option „Recalculate coordinates“	36
11.2	Menu option „Graphically“	36
11.3	Menu option „By hand“	37
11.4	Menu option „Page format“	37
11.5	Menu option „Font sizes“	37
12	Menu item Evaluation	37
12.1	General	37
12.2	Menu option „Normal isolines“	38
12.3	Menu option „Coloured“	39
12.4	Menu options „3D“ and „3D raster“	40
12.5	Menu option „Circles“	42
12.6	Menu option „Values in node section“	42
12.7	Menu option „Position“	42
12.8	Menu option „Any section“	43
12.9	Menu option „Position“	44
12.10	Menu option „Support“	45
12.11	Menu option „Principle stresses“	45
12.12	Menu option „Deformed grid“	46
12.13	Menu option „Animation“	46
12.14	Menu option „Rods“	47

12.15	Menu option „Flow“	47
12.16	Menu option „Mohr“	47
12.17	Menu option „Individual values“	48
13	Menu item ?	48
13.1	Menu option „Copyright“	48
13.2	Menu option „Maxima“	48
13.3	Menu option „Help“	48
14	Tips	48
15	Example.....	49
15.1	Step 1: Program preparation	50
15.2	Step 2: System input	50
15.3	Step 3: Boundary conditions.....	51
15.4	Step 4: Material characteristics.....	53
15.5	Step 5: Calculate slab.....	53
15.6	Step 6: Evaluate	54
15.7	Step 7: Drawing the stress bulb	55

1 Preface

The GGU-ELASTIC program can calculate 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. It is planned to extend the program to non-linear material laws. Owners of GGU-ELASTIC will then receive a favourable update offer.

The program system consists of a powerful grid generator and comfortable evaluation routines (isolines plans, 3D graphics, etc.) for calculation results.

Data input is in accordance with WINDOWS conventions and can therefore be learned almost without the use of a manual. Graphic output supports the True-type fonts supplied with WINDOWS, so that an excellent layout is guaranteed. Colour output and Bitmap graphics are supported. Using the integrated Mini-CAD program, you can also import DXF files (see the Mini-CAD manual, supplied with the program).

The following system minimum is needed to use the program:

- MS-DOS compatible computer (Pentium I processor or higher),
- WINDOWS '95, '98, 2000 or WINDOWS NT

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. 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 Installation

For installation and subsequent registration of GGU software, please refer to the information in the “Installation notes for GGU Software”, which are supplied the program.

3 Program concept

After this you may start the GGU-ELASTIC program according to WINDOWS conventions. The program is equipped with a large amount of „error“ questions. Even highly nonsensical input will usually be caught by the program and shown with an error message on the screen. Independently of this you should, for security reasons, intermittently save longer input to disk or hard drive, if only to save renewed input in the case of a power cut.

After starting the program you will see a start-up screen with a coordinate system.

The start-up screen shows two menu items at the top edge.

- File
- ?

After clicking the menu item „File” you can either „Load” an existing system or create a „New” system, using the appropriate menu options. After selecting the system, the upper edge of the window will show eight (ten) menu items:

- File
- FE grid
- Boundary
- System
- View
- Page
- Evaluate
- ?

After clicking on one of these, the so-called menu options fold down, from which you can then reach all program functions.

The program works on the principle of „What you see is what you get”. This means that the screen presentation represents, on the whole, that which you will see on your printer. With a consequent realisation of this principle, the screen 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 the [F2] key, or the [Esc] key. The [Esc] key will, additionally, set the screen presentation to A3.

4 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 calculate 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 calculate.
- Start the GGU-ELASTIC program and select the menu option „**File / New**”. Select the type of system.
- If necessary, fit the page coordinates to those of your system. For this, use the menu option „**Page / Coordinates by hand**”.
- Then select the menu option „**FE grid / Set 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 option „**FE grid / Edit nodes**”.
- If the nodes lie outside of the page coordinates, select the menu option „**Page / Recalculate coordinates**”.

- Then select the menu option „**FE grid / Grid by hand**” 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 option „**FE grid / Automatic grid**”.
- If you would like to edit the positions of grid nodes, select the menu option „**FE grid / Edit nodes**” or „**FE grid / Move nodes**”.
- If you would like to calculate a simple rectangular system, you can complete grid generation in a few seconds using the menu option „**FE grid / Raster**”.
- If you would like to delete a triangular element, select the menu option „**FE grid / Grid by hand**” again and click on the three corner nodes of the appropriate element. Using this menu option, try double-clicking in a triangular element.
- If you would like to assign different material characteristics to different triangles, use the menu option „**Boundary / Individual material**” or „**Boundary / In section**” to enter varying material numbers for individual or multiple elements. In the menu option „System / Material characteristics” 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 options „**FE grid / Individually refine**”, „**FE grid / Section**” or „**FE grid / All**”.
- Even after refining your grid you can edit the system as you wish by using „**FE grid / Set nodes**”, „**FE grid / By hand**”, etc.
- In order to get a numerically favourable FE grid, you should always select the menu option „**FE grid / Optimise**” and then select the sub-point „Diagonals”.
- For demonstration purposes you can create acute angled, and thus numerically unfavourable, triangular elements with the menu option „**FE grid / Move nodes**”. Then select the menu option „**FE grid / Optimise**” („Topology” button) and follow the effects on the screen.
- Define the decisive path boundary conditions for your system using, e.g., the menu option „**Boundary / Individual path**” or „**Boundary / In section**”.
- Define the decisive force boundary conditions for your system using, e.g., the menu option „**Boundary / Single loads**” or „**Boundary / Line loads**”.
- If necessary, edit the material characteristics using the menu option „**System / Material characteristics**”.
- If rod elements (e.g. piles or anchors) are present in the system, define the position of these elements via the menu option „**Boundary / Rods**”, by defining a „section” along the desired nodes. After pressing the [Return]-key you can assign the thus defined rods a material number. The rod stiffness’ (EJ and EF) can be edited using „**System / Material (rods)**”.
- Simple control of the defined boundary conditions is possible using the menu option „Boundary / Check”.
- When you have finished grid generation, select the menu option „System / Calculate”, and start the calculation. Before calculation begins a bandwidth optimisation will be carried out, if necessary, in order to achieve a numerically favourably configured equation system.
- After completion of calculations you can, if wished, print the results as a protocol or save them as a file. In general though, this type of result presentation is less than satisfactory.

- Instead, go immediately to the menu item „Evaluate”. Here you have a variety of evaluation possibilities. The menu option „Isolines, Coloured” is especially impressive, or the menu option „3D raster”. 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” switch 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 option „**File / Start**” and then selecting the „**Printer**” switch 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 calculations, please remember that all finite-element-methods are approximation methods. The quality of the approximation increases with increasing grid density. In the current version, systems with a maximum of 10 000 triangular elements and nodes can be calculated.

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 options need be selected for calculation. All further menu options are mainly for data saving, layout and, if necessary, further evaluation of calculations. A description will follow in the further chapters.

5 Theoretical principles

5.1 General

An analytical solution is only possible for simple systems. For calculation of complicated systems, one must rely on numerical solution procedures. In the main

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

are used. For finite procedures, the whole area is divided into many small (finite) areas (elements). Using FEM, triangles are usually selected for these area divisions. Within these triangles simple, generally quadratic, approximation functions are used. The actual, complicated solution is put together like a mosaic from the many simple partial solutions. Equation systems arise, the number of whose variables correspond to the number of system nodes. For finite difference methods one generally only has the possibility of defining a discrete whole using rectangular elements. In contrast to FEM therefore, FDM is a lot less flexible with reference to adapting to complicated boundary structures. It is also more difficult to carry out grid refinement in certain areas. Further to this, the resulting equation systems are numerically more stable for FEM. The main advantage of FDM consists of the theoretically less complex mathematical relationships, which will, in general, 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 methods or finite difference methods are approximation methods. The quality of the approximation to the actual solution increases with an increase in grid refinement. You should pay attention to the fact that the grid should be finer for areas in which the main forces act (e.g. single loads). The type of triangle also has a certain influence. Optimum conditions are achieved with isosceles triangles. An overview of the solution quality can be achieved by calculating the same system with a finer or coarser grid subdivision, and comparing the divergence of both solutions with one another.

The following general comments to the GGU-ELASTIC program are also 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, the number of whose 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 grid, 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 element grid 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 grid density. In the current version, systems with a maximum of 10 000 triangular elements and nodes can be calculated.

Note on 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 force or path boundary conditions, are automatically freely supported. In finite-element theory this type of boundary condition is also known as a natural boundary condition.

5.2 Signs and designations

The following sign rules are valid:

- displacement w_x and w_y e.g. in [m] positive in positive axis direction
- single loads P_x and P_y 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:

- σ_x = stress in x - direction
- σ_y = stress in y - direction
- σ_z = stress in z - direction (normal to plane of observation)
- σ_1 and σ_2 = principal stresses
- τ_{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).

- σ_r = stress in r - direction (horizontal)
- σ_z = stress in z - direction (vertical)
- σ_t = stress in t - direction (normal to plane of observation)
- σ_1 and σ_2 = principal stresses
- τ_{rz} = shear stress in plane of observation

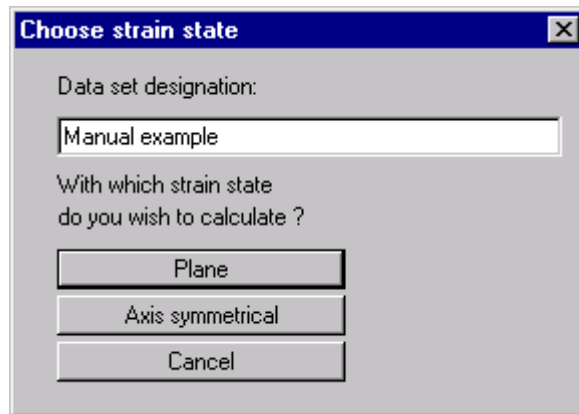
The program works with true dimensions. This means that you can freely choose the dimensions for force and length. If, for example, you decide on [kN] and [m], all input must be in these dimensions (e.g. then, Young's modulus in kN/m²).

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

6 Menu item File

6.1 Menu option „New“

After selecting the „New“ menu option, the type of system to be created must be determined in the following dialog box.



If a FE grid has already been entered, and you would like to switch from a plane to an axis-symmetrical system (or vice-versa) using the same grid, you can decide, after this dialog box, whether or not the current FE grid is to be kept.

6.2 Menu option „Load file“

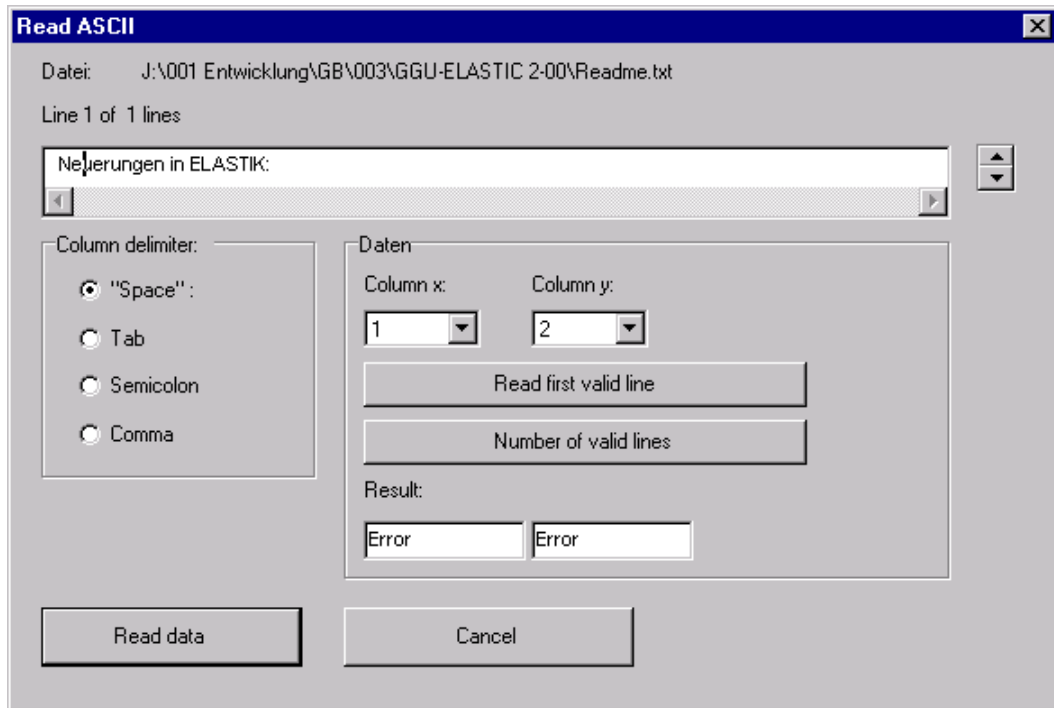
If you have entered data at a previous „sitting“ at the computer, and saved these as described in the menu option „Save“, you can reload them by selecting this menu option. It is then possible to edit these data according to your wishes.

6.3 Menu option „Save file“

You can save data entered during program use in a file, in order to have them available at a later date, or to archive them. For reasons of transparency, it makes sense to use „ELA“ as file suffix, as the file requester box for the menu option „Load file“ only shows files with this suffix. If you do not enter a suffix when saving, „ELA“ will be used automatically. If the current system has already been calculated, you can save the results with the data set. Upon reloading this file, the results are available for evaluation without having to be recalculated.

6.4 Menu option „Read ASCII file”

If coordinates of FE grid nodes are available in the form of an ASCII file, they can be read into the program. The data must contain, in each row, the x- and the y-value for each node. Decimal fractions must be shown by a point (not a comma). When loading the ASCII file, you must indicate in which columns the x- and y-values are to be found.



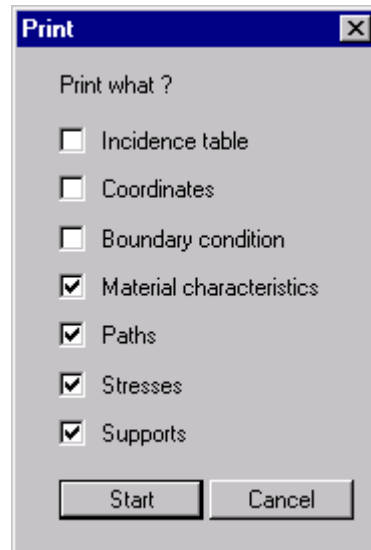
The current line of the ASCII file is shown at the top. Using the arrows at the right you can move through the file. If all input is correct, the result for this line will appear in the box below the columns. Otherwise „error“ will appear. You may then have to alter the column delimiter. If the file contains invalid as well as valid lines, these will simply be skipped when reading. Finally, select the „Read data” button. You can then further process the read-in coordinates to a FE grid.

6.5 Menu option „Save ASCII file”

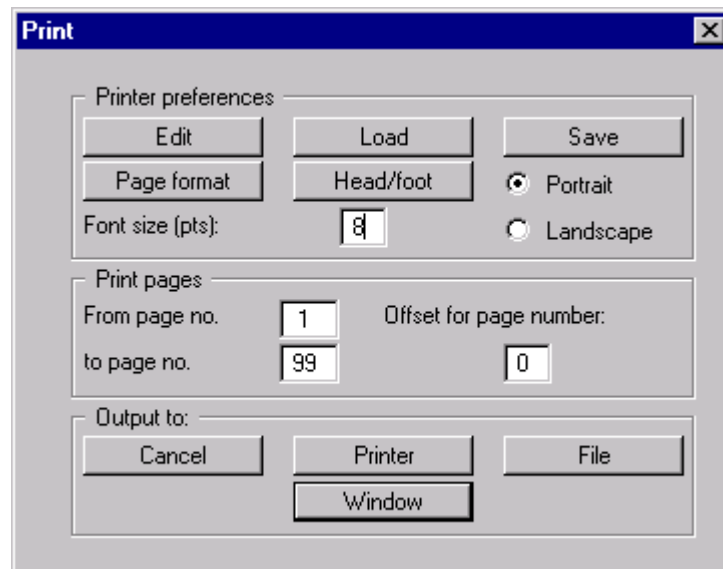
When a finite-element grid has been generated, the node coordinates can be saved in an ASCII file, to enable them to be used in non-GGU programs.

6.6 Menu option „Protocol output“

You can have a result protocol sent to the printer or to a file (e.g. for further processing with a word processor). Output contains all information on calculations, including system data. First, decide how comprehensive output is to be



by activating the appropriate switches.



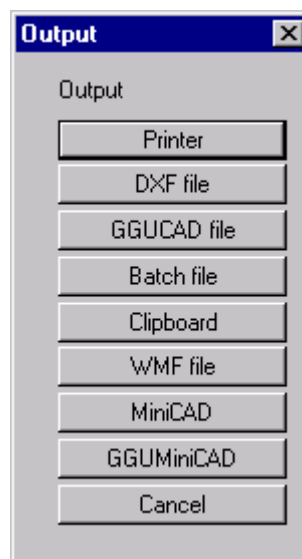
You can then set output preferences. With the „Edit“ button, you can edit the current printer preferences or select a different printer. With the „Save“ button, you can save all preferences from this dialog box in a file, in order to have them available at a later sitting. If you select „ELASTIK.DRK“ as file name and save the file on the program level (default), the file will be automatically loaded at the next program start. With the „Page format“ button you can define, amongst others, the size of the left margin and the number of lines per page. With the „Head/foot“ button you can enter a header and foot text for each page. If the „#“ sign appears within the text, the current page number will be entered during printing (e.g. ‘Page #’). If page numbering is not to begin at page „1“, an offset can be defined. This offset will be added to the current page number. The text size can be entered in pts. With the buttons at the bottom of the dialog box, output is sent to the „Printer“, to a „Window“ or to a „File“, the name of which must then be entered. If you select the „Window“ button, the results will displayed in an additional window. In this window editing is possible, as well as loading, saving and printing of text.

6.7 Menu option „Printer preferences“

You can edit printer preferences or change printer in accordance with WINDOWS conventions.

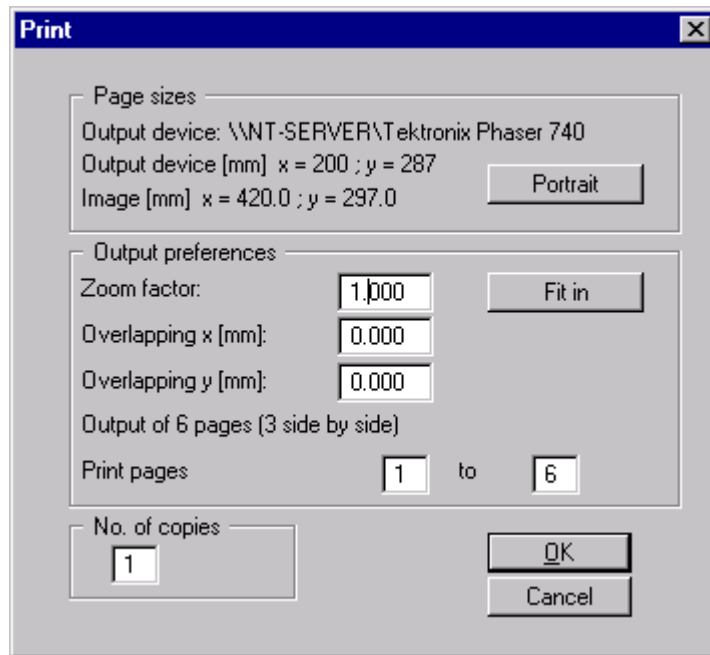
6.8 Menu option „Start“

The following dialog box appears:



„Printer“

allows graphic output of the current screen contents to the WINDOWS standard printer or to a different printer, selected in the menu option „Printer preferences“. For direct output, the following dialog box appears:



In the upper part of the dialog box, the maximum dimensions which the printer can accept are given. Below this, you can see the dimensions of the image to be printed. If the image is larger than the output format of the printer, the image will be printed to several pages (in the above example, 6). In order to be better able to re-connect the images later, 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 in”). Following this, you can enlarge to the original format on a copying machine, to ensure a true scaling.

„AutoCAD file / GGUCAD file“

allows output of the current screen contents to a file, in order to further process the image in a different program (e.g. AutoCAD, GGUCAD). AutoCAD output is in the so-called DXF format, which is standardised. With reference to the DXF format, please see the note in the menu item „Preferences/General“, menu option „WINDOWS font” (see 8.2). Bitmap graphics and colour fill will not be exported to AutoCAD.

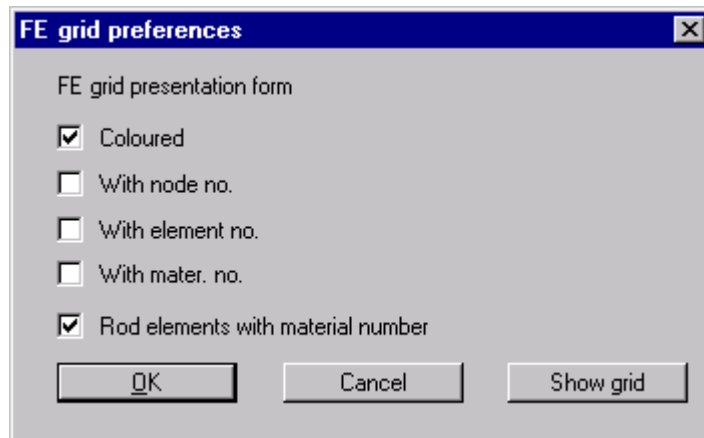
„Clipboard“

The current screen contents are copied to the WINDOWS clipboard. From there, they can be imported into other WINDOWS programs for further editing, e.g. into a word processor. Use of the Metafile format guarantees the best possible quality when transferring graphics.

7 Menu item FE grid

7.1 Menu option „Preferences“ and „FE grid“

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



The switches determine what is displayed on the screen with the FE grid, after calling-up the „FE grid“ menu option. The element number and material number cannot be displayed simultaneously.

If the screen display no longer shows the FE grid, select the menu option „Page / Recalculate coordinates“, or press the [F9] key.

7.2 Menu option „Outline“

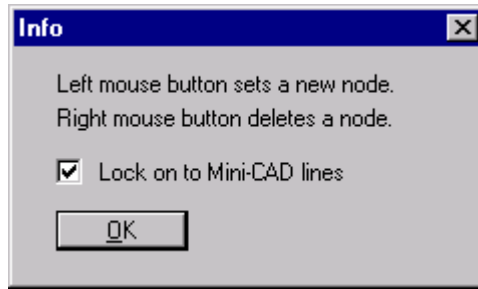
After selecting this menu option the outlines of the various soils in the FE system, as defined in „FE grid / Preferences“, will be displayed.

7.3 Menu option „Set nodes“

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 option can also be reached with the [F3] function key.

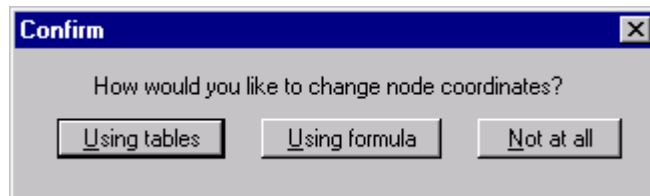
When setting new nodes, the position in x and y direction is shown in the upper window bar. If you have a scanner at your disposal, you can scan in the system to be processed and save this as a so-called Bitmap file (suffix *.BMP). This Bitmap can be displayed on the screen using the Mini-CAD program module (see separate Mini-CAD manual). This simplifies further input of important system nodes.

Alternatively, you can also import a DXF file, which contains, amongst others, the system contours, using Mini-CAD (see Mini-CAD manual). If Mini-CAD data are present, a switch appears in the initial dialog box:



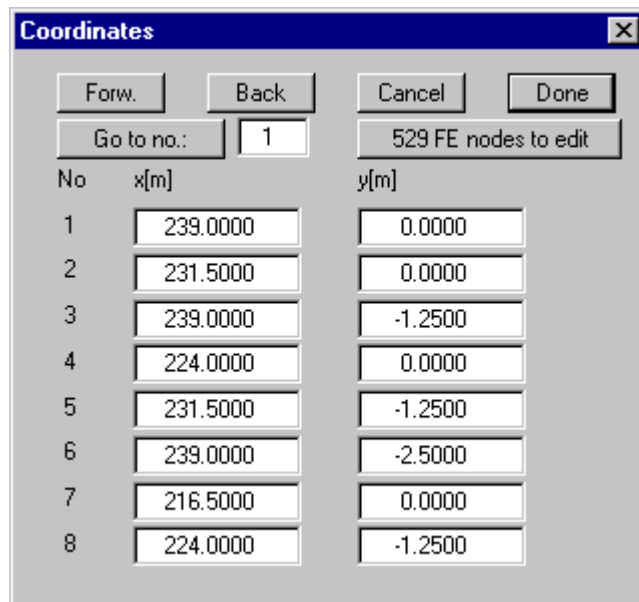
If you activate this switch, the mouse cursor appears as a rectangle with cross-hairs. If a Mini-CAD line end is within this rectangle, the program will lock exactly on to this point or, if several points are within the rectangle, on to the one nearest to the centre point.

7.4 Menu option „Edit nodes“



Using tables:

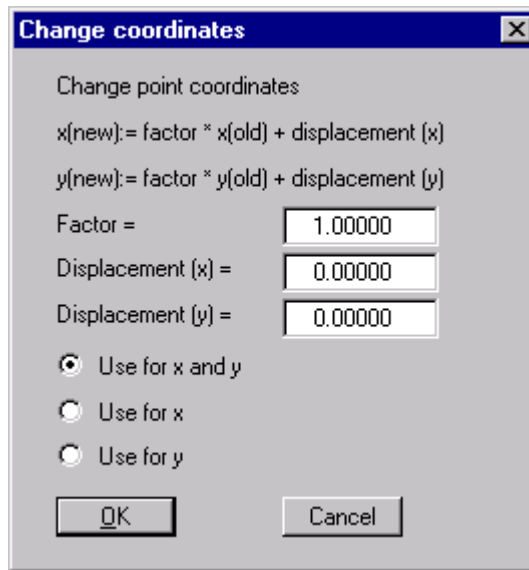
Using this menu option you can edit the system nodes previously entered with the mouse or, alternatively, enter new node coordinates for a new system.



If you would like to edit the number of nodes, select the „x FE nodes to edit“ button and then enter the new number of nodes. Using „Forw.“ and „Back“ you can move through the list.

Using formula:

If you have entered the coordinates in the wrong scale, you can „repair” them using this menu option.



7.5 Menu option „Move nodes“

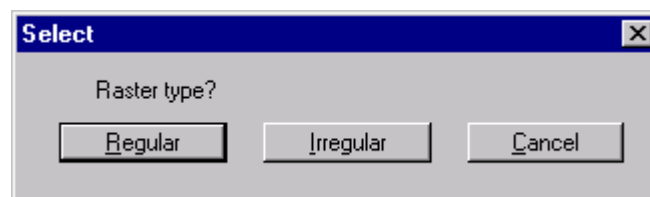
After selecting this function, the defined FE system with the FE elements appears. Each node can be moved with the left mouse button pressed. At the top of the window bar the current coordinates of the nodes are shown. The last node displacement can be undone using the Backspace-key.

7.6 Menu option „Edit nodes“

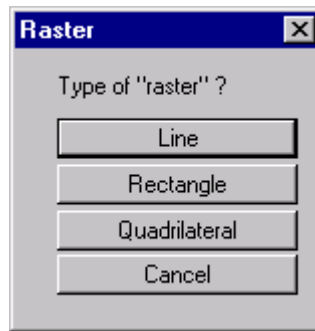
By double-clicking on a node with the left mouse button a dialog box appears in which the coordinates can be edited via keyboard.

7.7 Menu option „Raster“

First, decide upon:



7.7.1 „Regular“

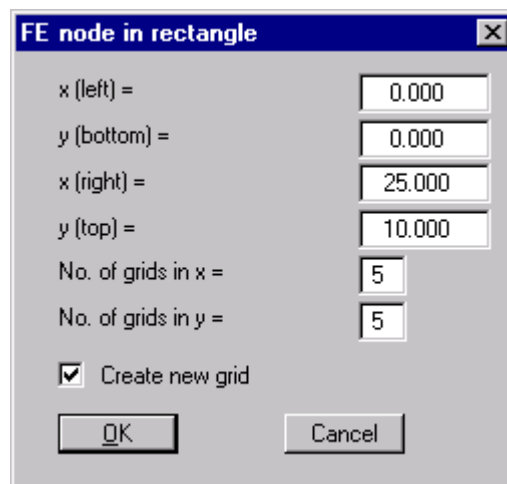


Alternatively, you can also set nodes

- along one or more lines
- in one or more defined rectangles
- in one or more defined rasters

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

„Rectangle“



You enter the corners of the raster and the number of subdivisions. If one of the thus created node points lies within an existing FE grid, and the „Delete current grid“ switch is not activated, the node points created will not be used.

If you activate the „Create new grid“ switch (only works when the „Delete current grid“ switch is activated), a FE grid will be created together with the FE nodes.

7.7.2 „Irregular“

With the raster method in Section 7.7.1, the nodes have the same spacing in each row created. With „irregular“ this spacing can be varied.

dx values		dy values		Recalculate x and y values	
No. of dx values		No. of dy values			
0.000	dx [m]	0.000	dy [m]	x0 =	0.00
1.000	1.000	1.000	1.000	y0 =	0.00
2.000	1.000	2.000	1.000	<input type="checkbox"/>	Delete current mesh
3.000	1.000	3.000	1.000	<input type="checkbox"/>	Create new mesh
4.000	1.000	4.000	1.000	OK Cancel	
5.000	1.000	5.000	1.000		

With the „No. of dx-values“ and „No. of dy-values“ buttons, you can define the number of rasters. For „dx“ and „dy“ you then enter the raster spacing. If you press the button „Recalculate x and y values“, the first column at the left of „dx“ and „dy“ will be recalculated. These are the raster absolute values. For „x0“ and „y0“ you enter the raster origin. If one of the thus created node points lies within an existing FE grid, and the „Delete current grid“ switch is not activated, the node points created will not be used.

If you activate the „Create new grid“ switch (only works when the „Delete current grid“ switch is activated), a FE grid will be created together with the FE nodes.

7.8 Menu option „By hand“

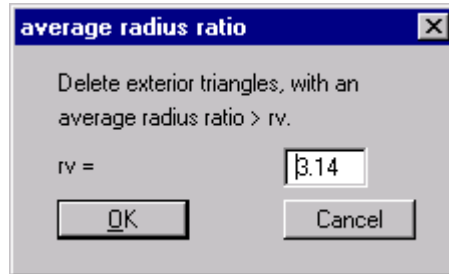
After grid node input, a finite-element grid is defined with this menu option. You must click on three nodes with the left mouse button. After selecting three nodes a box appears, in which the material number for the FE element must be defined. This menu option can also be reached with the [F4] function key. A finite-element can be deleted or edited by clicking on the three corresponding nodes with the left mouse button once again.

7.9 Menu option „Automatic“

After grid node input, automatic grid generation can be carried out with this menu option (Delauney triangulation). If a FE grid already exists, the current grid can be deleted or supplemented. When supplementing a current FE grid, „air holes“ can result (not completely filled FE grid), if the current grid was not created using Delauney triangulation. Such areas must be reworked by hand, or the complete grid must be re-triangulated. All newly created triangles are given the material number „1“.

7.10 Menu option „Round off“

With Delauney triangulation a triangular grid is created which surrounds all nodes. In this way triangular elements can occur which, in the boundary areas, can be very acutely angled. Using this menu option you can remove such triangles from the FE grid.



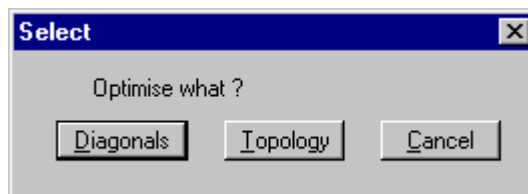
The radius ratio describes the relationship between exterior radius and interior radius of a triangle. For an isosceles triangle this ratio equals 2.0 (optimum). Before you arrive at the dialog box above, you will be shown the maximum and the average radius ratio of the FE grid. In the example above, all exterior triangles with a radius ratio larger than 3.14 will be removed. To avoid chaos, only exterior triangles will be removed.

7.11 Menu option „Delete“

With this menu option you can delete selected system. You must click on four points in an anti-clockwise direction. All triangles with a centroid within the quadrilateral will be deleted.

7.12 Menu option „Optimise“

First, enter the type of optimisation:



7.12.1 Optimise diagonals”

Optimisation of diagonals is implemented in order to create numerically favourable FE grids, i.e. where possible, isosceles triangles. The effects of optimising diagonals can best be shown using an example:

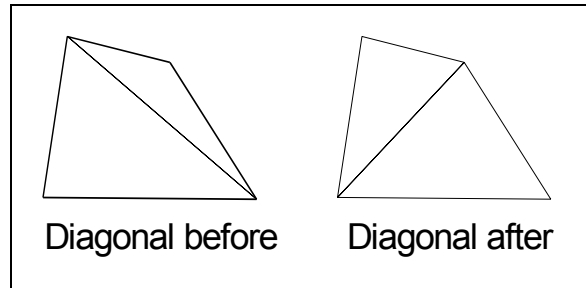


Figure 1 Optimisation of diagonals

If a current, „unfavourable“, diagonal cuts two different material areas, no optimisation takes place, as this would alter the system.

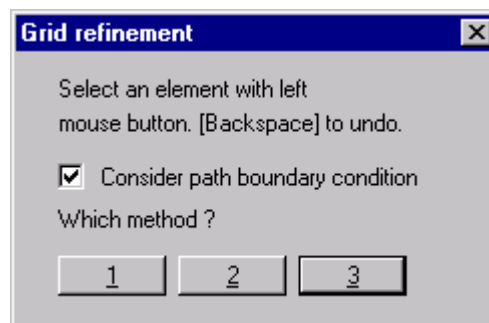
7.12.2 Optimise topology

With this program routine, the triangular element nodes are displaced in such a manner as to create, as far as possible, isosceles triangles. Isosceles triangles have especially favourable numerical properties. As the displacement of system boundaries, and element boundaries with neighbouring elements which consist of different materials, is not sensible, these element boundaries will be fixed from the outset. Nor will nodes be moved at which path boundary conditions have been defined. Optimisation of the FE grid can be followed on the screen by setting the „With graphics“ switch. The optimisation can be cancelled at any time by pressing the right mouse button.

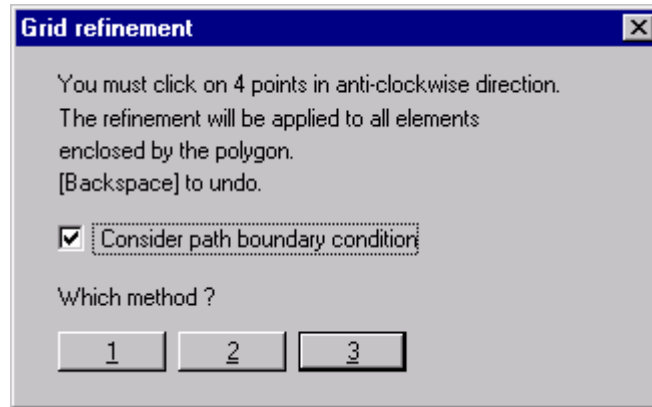
7.13 Menu option „Refine“

The grid elements can be refined with the following functions.

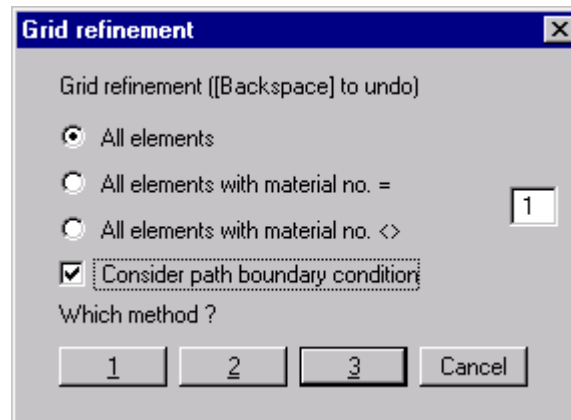
- „Individually“
After selecting this menu option the following box appears:



- „In section“
After selecting this menu option the following box appears:



- „All“
After selecting this menu option the following box appears:



Upon activating the „Consider path boundary condition” switch, new nodes which lie immediately between two nodes with path boundary conditions will be assigned the average of the two values. For „force boundary conditions” this procedure is not unambiguous, and can lead to misunderstandings. Force boundary conditions will therefore not be „refined”.

Three different methods can be used for element refinement. Refinement will be shown on the following grid using the individual elements 9 and 13 as examples.

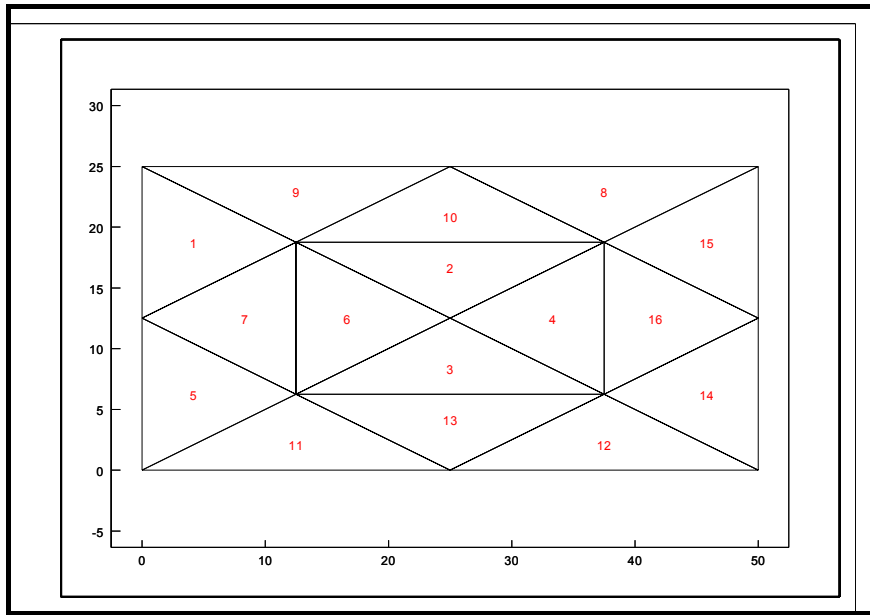


Figure 2 Basic grid for exemplary grid refinement

7.13.1 Method 1

In the centroid of the selected triangle an additional node is created.

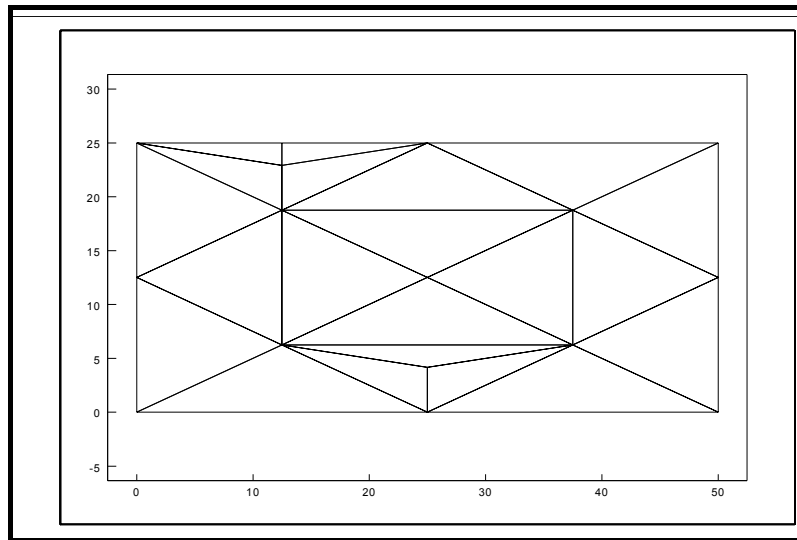


Figure 3 Method 1 for grid refinement

7.13.2 Method 2

The selected triangular element and the neighbouring element are halved.

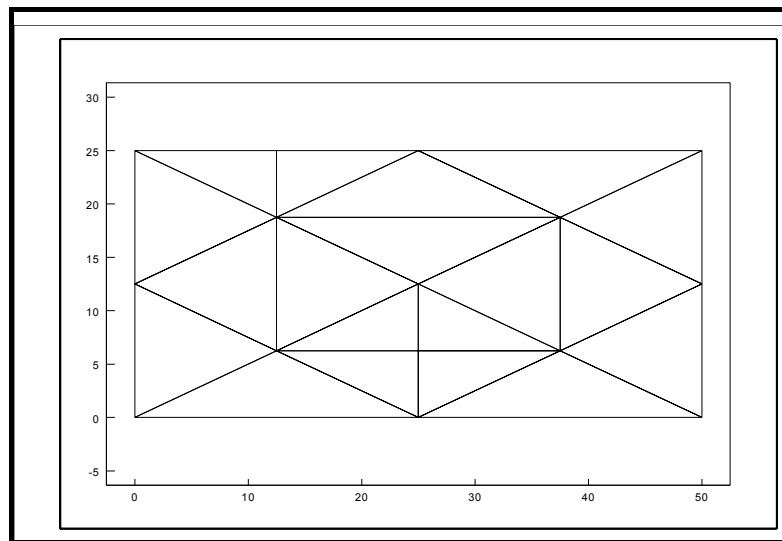


Figure 4 Method 2 for grid refinement

7.13.3 Method 3

A new triangular element will be inserted at the median of the clicked triangular element. The neighbouring triangular elements will be halved.

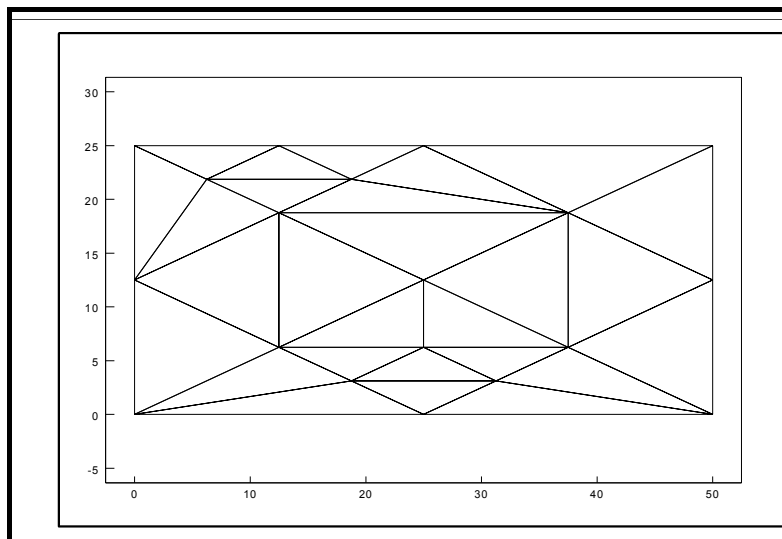


Figure 5 Method 3 for grid refinement

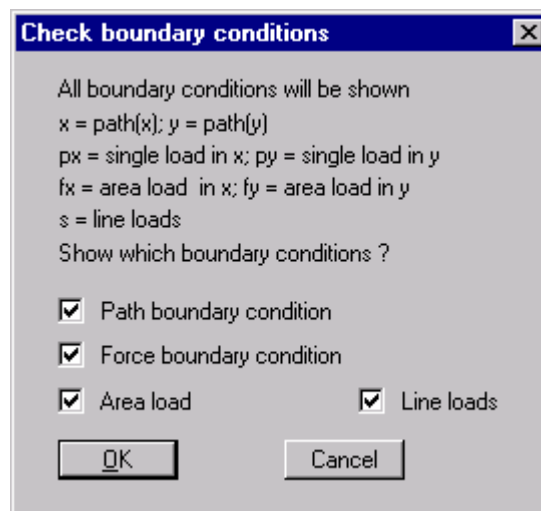
8 Menu item Boundary

8.1 Menu option „Preferences”

In the following dialog box preferences for the presentation of the boundary conditions on the screen can be set.

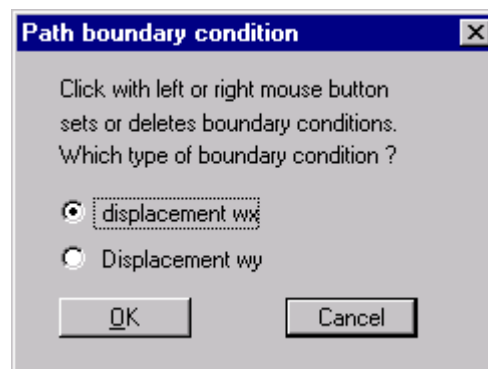
8.2 Menu option „Check”

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



8.3 Menu option „Individual path”

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

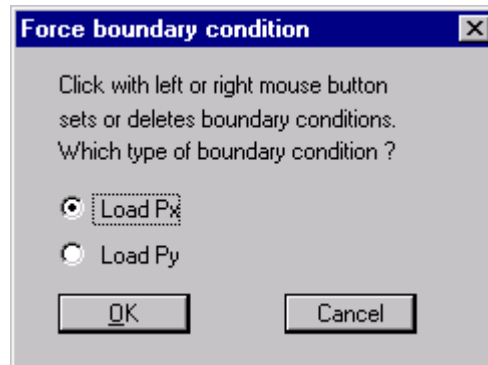


8.4 Menu option „In section”

After selecting this menu option all nodes in a section can be assigned a path 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.

8.5 Menu option „Single loads“

The dimension of single loads is e.g. kN/m. With this menu option the single loads in the FE system are agreed upon, in analogy to the path boundary conditions.



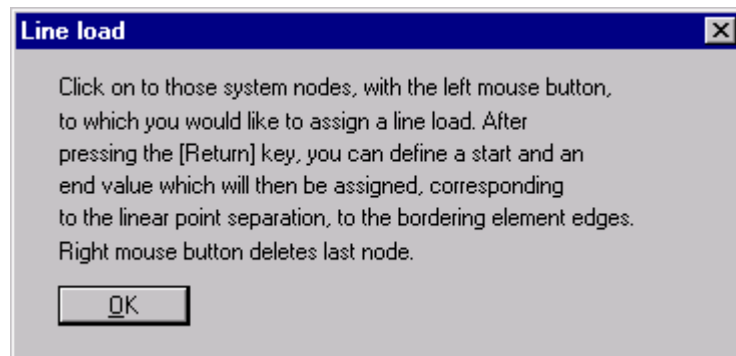
Note on free supports:

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 path or force boundary conditions, are automatically freely supported.

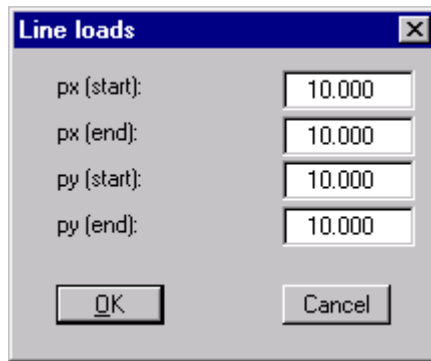
8.6 Menu option „Line loads“

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

For a general description see the menu option „Single loads“. The procedure for definition of line loads is explained in the dialog box.

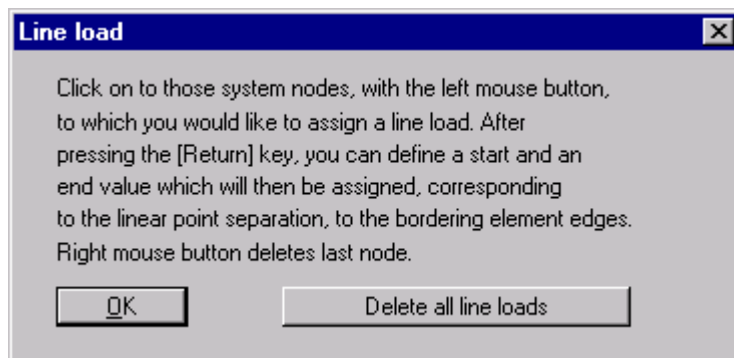


After clicking on the system nodes and confirming with the Return-key

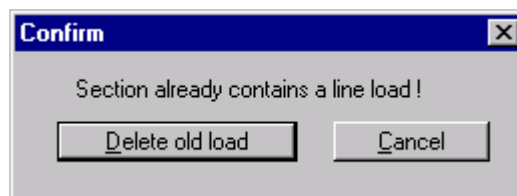


the intermediate boundaries will be assigned the appropriate value.

If line loads have already been defined an additional button appears in the above dialog box, with which current line loads can be deleted.



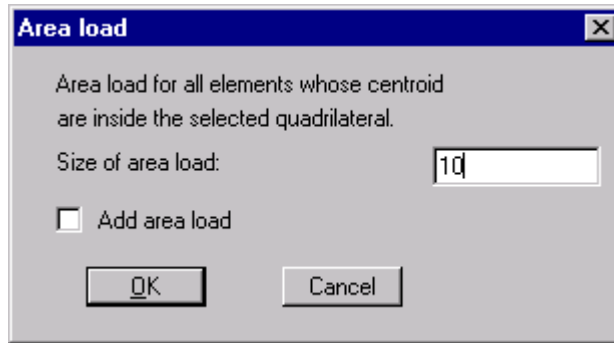
Individual line loads can be deleted by reselecting them. The following dialog box then appears:



8.7 Menu option „Area loads”

The dimension of area loads is e.g. $\text{kN/m}^2/\text{m} = \text{kN/m}^3$.

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 option „System / Material characteristics“. Area loads in y-direction are rather rare in soil mechanics. An area load in y-direction would be e.g. a centrifugal force.



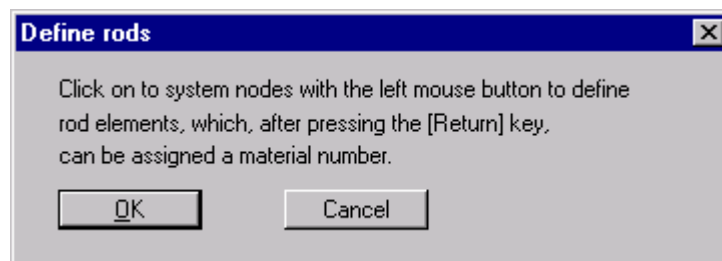
If you select the „Add area load” switch, the value entered will be added to any area loads already present.

8.8 Menu options „Individual material” and „In section”

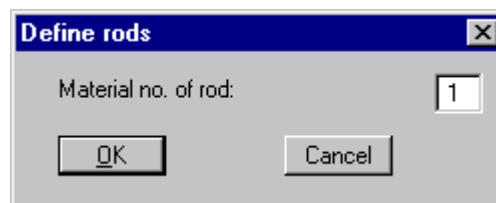
With these two functions you can edit triangular element materials, after selecting the screen presentation form for the FE system. Depending upon the number of materials selected, you can later edit the material characteristics (Young’s modulus, ν , γ , etc.) in the menu option „System / Material characteristics”, individually for each material.

8.9 Menu option „Rods”

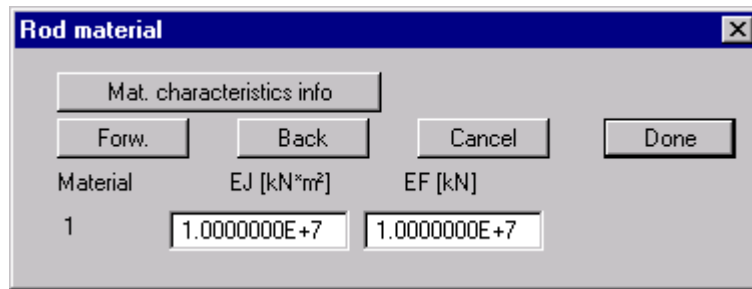
After you have defined a FE grid, you can define rods along nodes of the FE grid which can later be considered when calculating stiffness. This is useful e.g. when taking the influence of piles or similar objects into account.



After you have clicked on the nodes a dialog box appears,



in which you can assign a material number to the defined rods. 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 rod materials, the same number of lines appears in the menu option „System / Material (rods)” for definition of rod materials.



The defined rods are shown in the graphics with the appropriate material number. The colour and line strength can be edited in the menu option „View / Pens”.

If you would like to delete certain rods, click on these rods once again.

8.10 Menu option „Delete all“

You can delete all defined rods.

9 Menu item System

9.1 Menu option „Info“

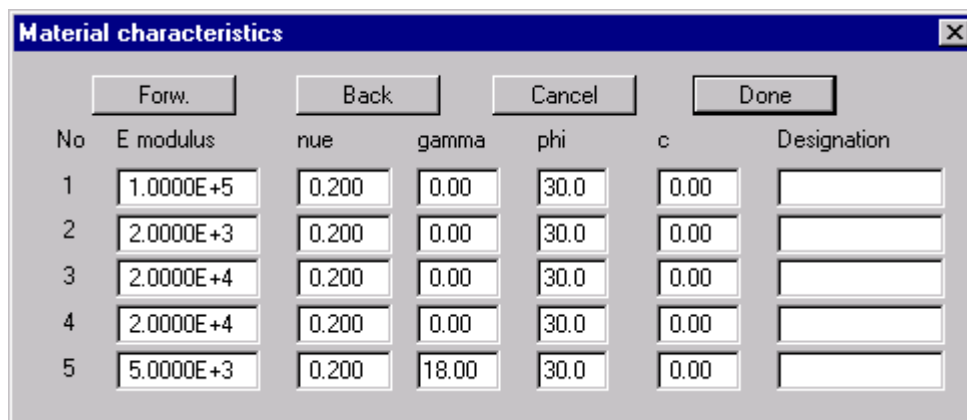
After selecting this menu option information will be shown on the selected system, with the number of triangular elements, nodes and boundary conditions.

9.2 Menu option „Data set designation“

You can enter a description of the current system. This text appears in the „General” legend.

9.3 Menu option „Material characteristics“

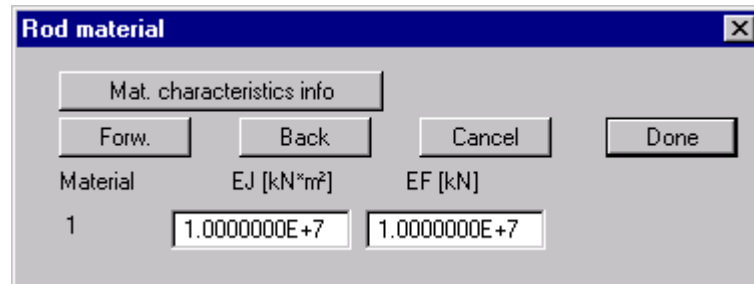
You can edit the characteristics. A dialog box appears with a number of input lines dependent on the number of materials.



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

9.4 Menu option „Material (rods)“

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



The following designations are valid:

- EJ = rod bending resistance
- EF = rod elasticity coefficient
- E = modulus of elasticity of rod
- J = moment of inertia of rod
- F = rod area

9.5 Menu option „Test system“

The system is tested for overlapping of triangular elements. The test can be cancelled at any time by pressing the right mouse button.

9.6 Menu option „Calculate“

After selecting this menu option the bandwidth of the equation system is optimised.

NOTE ON BANDWIDTH:

With finite-element methods equation systems arise which, depending on the system, only have values different to „0“ in the region of the principle diagonals. The iterative equation solver contained in FE programs uses this fact in order to save memory and thus shorten calculation time. The maximum distance between the principle diagonal and the „furthest non-zero element“ of the equation system is known as the bandwidth. In principle, iterative equation solvers do not require bandwidth optimisation. However, the results are much more stable and the calculation time shorter if the bandwidth is small.

After bandwidth optimisation is complete, the following dialog box appears:



After pressing the „Start” button you will be informed as to the current state of calculations. After calculations are complete



you can evaluate the results.

10 Menu item View

10.1 Menu option „Refresh”

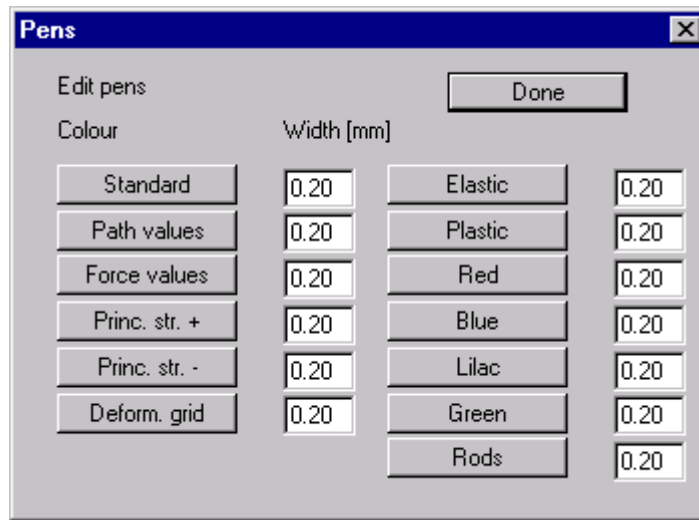
If, e.g., after using the zoom function (see below), only part of the image is visible, you can achieve a complete overview using this menu option. The zoom factor is definable. It is much simpler, however, to get a complete overview (zoom factor = 1.0) using the [Esc] key. With the [F2] key you can refresh the screen without altering the zoom factor.

10.2 Menu option „Zoom”

You will see information on the zoom function.

10.3 Menu option „Pens”

You can adjust the colour presentation of lines and lettering to suit your needs.



Click e.g. on the „Rods” button to edit the colour of any rods present. With width you enter the width of the pen in mm.

10.4 Menu option „Font”

With this menu option you can select a different font for the presentation Menu option „WINDOWS font”

For output to the screen or to an output device the program uses the WINDOWS specific true-type fonts, which guarantee an excellent presentation layout. In certain cases, however, it may be desirable to switch to a vector font. The vector font STANDARD.STZ is delivered with the program. You can switch to this font by clicking on this menu option. Clicking again switches back to the WINDOWS font. It is useful to switch to the vector font if you would like to create an AutoCAD (DXF format) file (see also menu option „File / Start”), as AutoCAD does not support WINDOWS fonts, and the file will then contain lines only.

10.5 Menu option „Mini-CAD and CAD for header data”

With these two menu options you can add free text to your graphics or equip them with additional lines, rectangles and bitmaps. For details please see the supplied „Mini-CAD” manual. The same dialog box appears for both menu options, the functions of which are further explained in the „Mini-CAD” manual. There are the following differences between „Mini-CAD and CAD for header data”

- Drawing elements created with Mini-CAD are with reference to the system coordinates of the drawing, and will be displayed accordingly. You should therefore always select this menu option, when you wish to enter additional information in the system area. All information in these Mini-CAD data will be saved with the file and reloaded when the file is opened the next time.

- Drawing elements created with CAD for header data are with reference to the page format (in [mm]). They therefore always remain at the same position on the page, regardless of the coordinates of the system. You should always use this menu option when entering general information (e.g. company logo, report number, annex number, stamp). When you save these so-called header data (see Mini-CAD manual), they can be reloaded into a completely different system (with different system coordinates). The thus saved header data will once again be at the same position on the page. This much simplifies creation and management of general page information.

10.6 Menu option „Smarticons”

After program start-up a pop-up window appears in the top right of the program window.

By clicking on these symbols (Smarticons) you can directly reach most of the program functions. If you would rather work with a horizontal bar, you can carry out the necessary alterations using this menu option. You can also switch off the Smarticons. The preferences will be saved, amongst others, in the „ELASTIK.ALG” file (see menu option „Save preferences”), and will be active at the next program start.

The meaning of the Smarticons will appear in the program window bar if you hold the left mouse button pressed over the symbol.

10.7 Menu option „3D Smarticons”

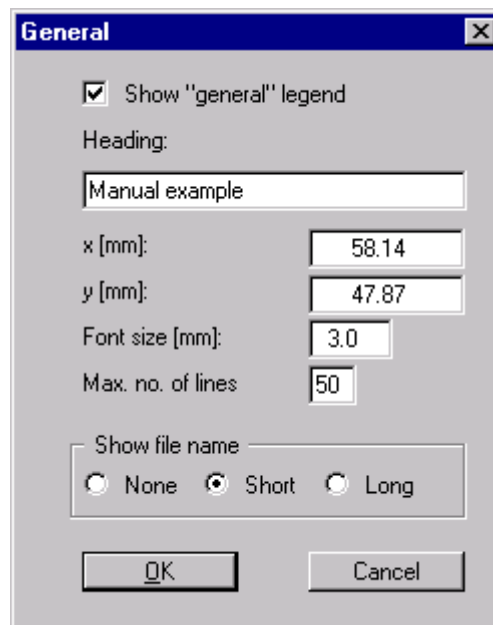
For 3D-graphics a pop-up window appears at the bottom left with 9 icons, with which 3D-graphics can be rotated and zoomed.

The meaning of the Smarticons will appear in the program window bar if you hold the left mouse button pressed over the symbol.

Using this menu option you can decide whether or not the pop-up window is to be displayed for 3D-graphics.

10.8 Menu option „General legend”

You can have a legend with general information displayed.

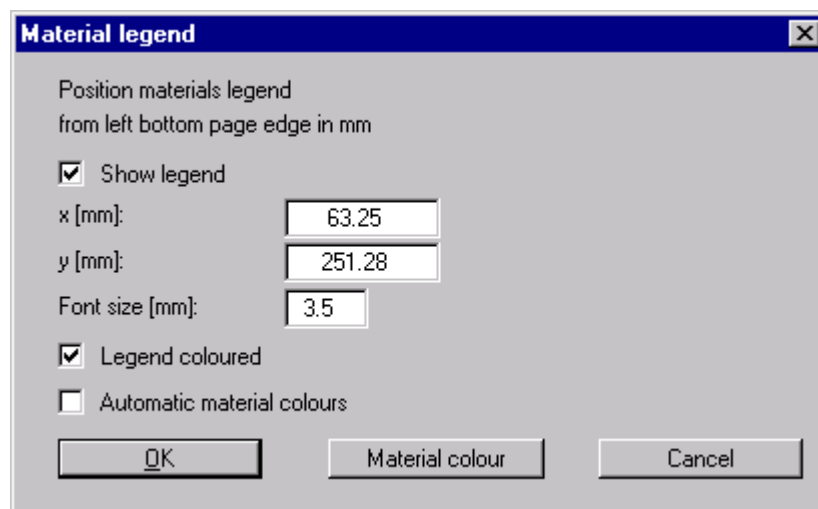


This legend contains information on the current graphics, as well as on heading and data set designation. If e.g. you have the velocity field displayed, you will see information on the velocities scale.

The x and y values define the position of the legend. If the legend contains several lines you can force a multi-column presentation with the appropriate settings in „Max. no. of lines“. Additionally, the file name can be shown in the legend.

10.9 Menu option „Materials legend”

You can have a legend with material characteristics displayed.



The x and y values define the position of the legend. If the „Legend coloured „ switch is activated, coloured boxes will be displayed instead of material numbers. The material colours can be edited with the button „Material colour“. The colours used here will also be used for a colour presentation of the FE grid, if the „Automatic material colours“ switch is deactivated. Otherwise, colours will be selected by the program.

10.10 Menu option „Rod legend“

You can have a legend displayed, with general information on any rods present.

10.11 Menu option „Move legends“

It is a chore to position the legends via x and y value input. If you select this menu option positioning may be carried out with the mouse. If you press the [F11] function key, the menu option will be selected without the descriptive dialog box.

10.12 Menu options „Save and Load preferences“

Saves all preferences selected with the menu item „View“. If you use the file name „ELASTIK.ALG“ the program will use these preferences at the next program start.

11 Menu item Page

11.1 Menu option „Recalculate coordinates“

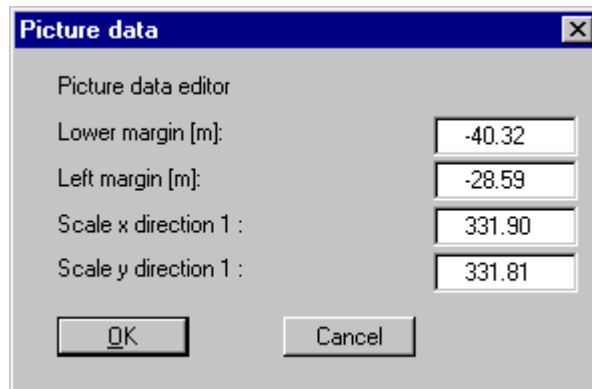
If you have altered the image coordinates during use of the program, e.g. by creating sections, you can the coordinates for a complete overview recalculated using this menu option. This function can also be reached with the [F9] function key.

11.2 Menu option „Graphically“

You can alter the image coordinates graphically with the mouse. An info box explains the possibilities. You can get new section coordinates by holding the [CTRL]-key, [Shift]-key and the left mouse button, and marking an area.

11.3 Menu option „By hand“

You can alter the image coordinates by direct numerical input.



11.4 Menu option „Page format“

When starting the program the default page setup is A3. If you would prefer different page formats you can define them in this dialog box.

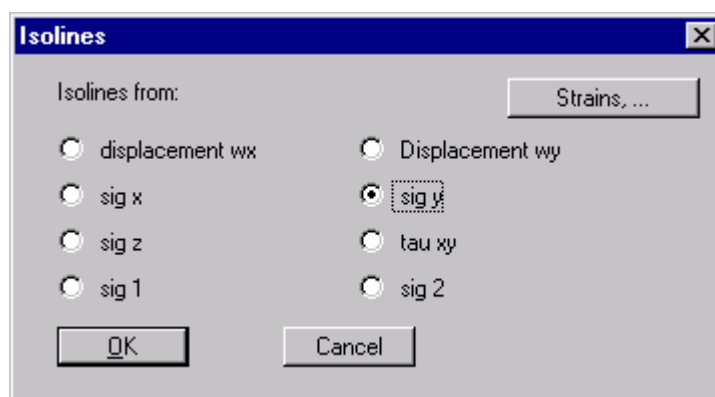
11.5 Menu option „Font sizes“

You can edit font sizes for image labelling.

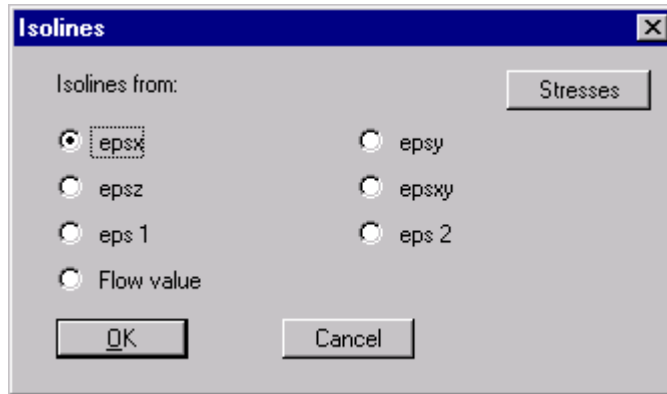
12 Menu item Evaluation

12.1 General

A variety of possibilities are available for evaluation and graphical presentation. Before every isoline presentation a dialog box appears.



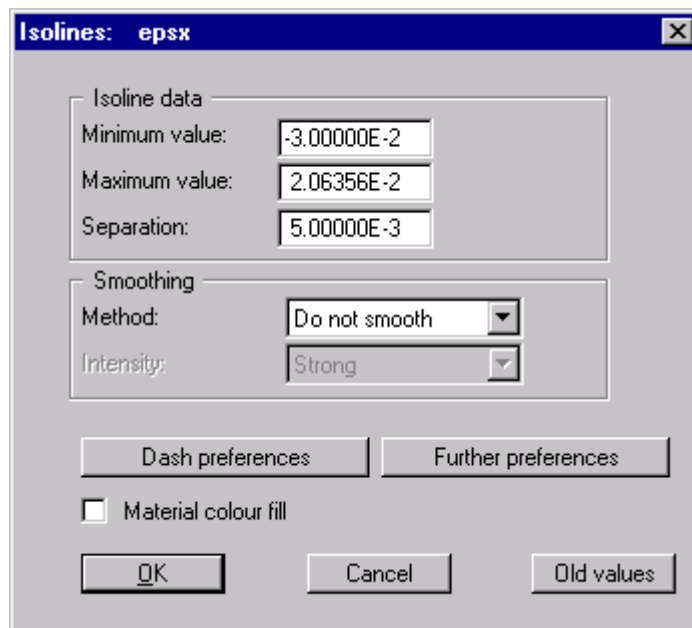
Here you can select the state variables to be displayed. If you would like to have strains displayed, click on the „Strains,..“ button and you will see:



If you select the „Stresses“ button, you will once again see the dialog box for selection of strains and displacements (see above).

12.2 Menu option „Normal isolines“

With this menu option you can have lines of equal state variables displayed. In the dialog box which appears you can influence the type of presentation.



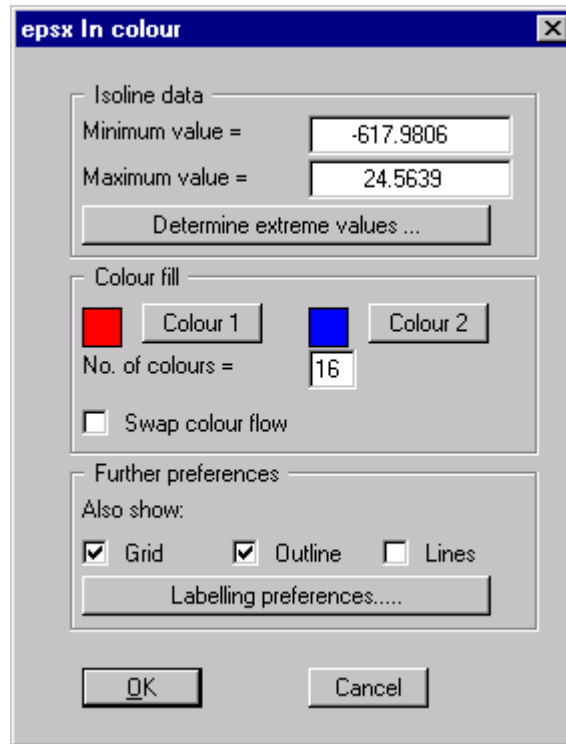
Further to this, you may select from three smoothing methods:

- Do not smooth
As a linear method is implemented within the triangular elements, you will get calculation results without program-internal smoothing.
- Method 1 and Method 2
With these smoothing methods two different Bezier splines are used to get smoothed isolines. Further control is possible via the smoothing number (0 – 3). Method 2 creates very „round“ isolines, with the danger that the true results may be falsified.

With the „Further preferences“ button you can edit the labelling etc. of the isolines. With the „Dash preferences“ button you can also have dashed or dash-dotted isolines created, similar to the contour lines known from cartography.

12.3 Menu option „Coloured“

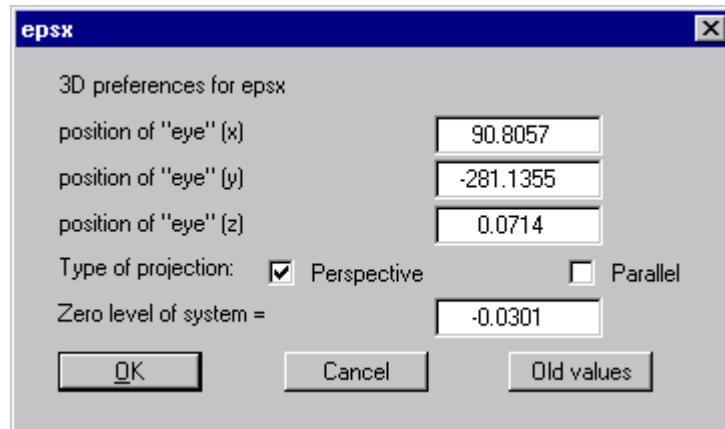
With this menu option you can have lines of equal state variables displayed in colour.



A colour flow from colour 1 to colour 2 will be created. The resolution can be set by the number of colours used. After selecting the boxes „Colour 1“ and „Colour 2“ you can set the start and end colours to suit your wishes. If the „Swap colour flow“ switch is activated, the colour flow will be from „Colour 2“ to „Colour 1“. You are advised to click on the „Determine extreme values“ switch before presentation. The minimum and maximum value of the state variable will thus be calculated and entered into the appropriate boxes. Naturally, you may enter this value via the keyboard.

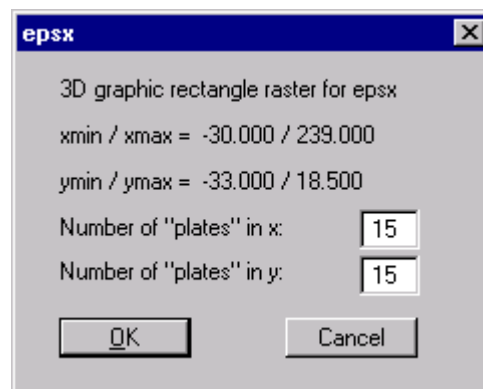
12.4 Menu options „3D” and „3D raster”

With this menu option 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”.

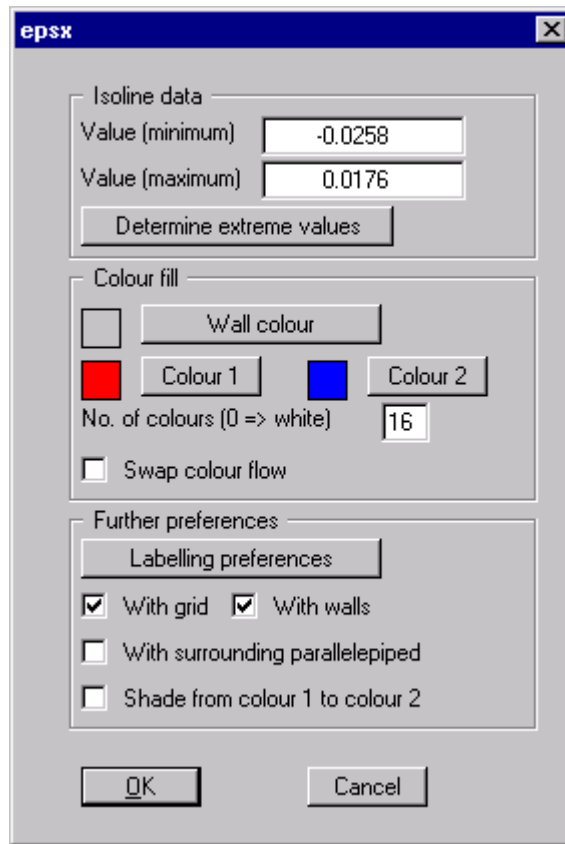


In general, the program will make a sensible suggestion for the input values in the dialog box, so you will not normally need to make alterations.

For irregular triangular grids, the 3D impression is generally lost. In such cases you are advised to use the menu option „3D raster”.



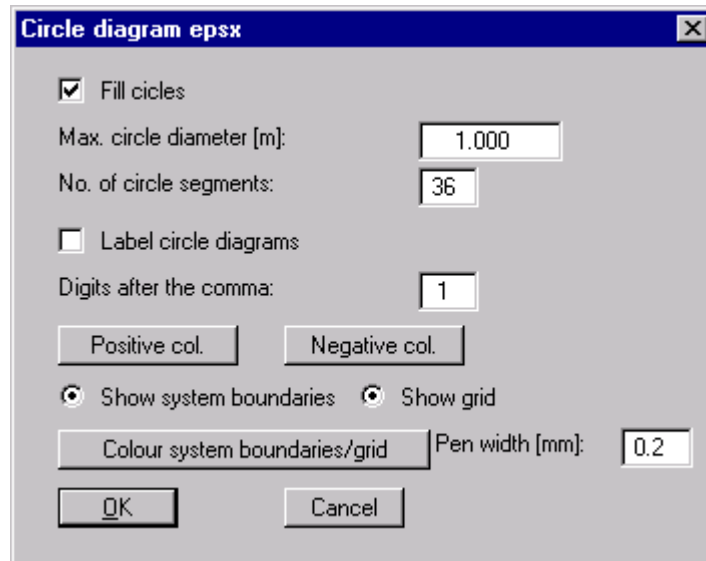
The program calculates the state variables at the raster points by linear interpolation of the FE calculation results. You will then see the following dialog box:



On the whole, the dialog box is the same as that for the menu option „Coloured“. Additionally, you can decide whether a surrounding parallelepiped is to be displayed which, in some cases, improves the 3D impression. A speciality is the switch „Shade from colour 1 to colour 2“. When this switch is activated, a light source will be simulated in the eye area. The angle between the light ray and the 3D surface is a measure of the reflection. Shading will be in accordance with colours 1 and 2. A useful colour choice would be e.g. dark grey for colour 1 and light grey for colour 2.

12.5 Menu option „Circles“

You can create circle graphics in which the appropriate state variable is shown as circles of different sizes (value-dependent).



If you activate the „Fill circles“ switch the circles with positive values 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 FE grid can be shown. With „Colour system boundaries/grid“ and pen width you can set preferences for graphical presentation.

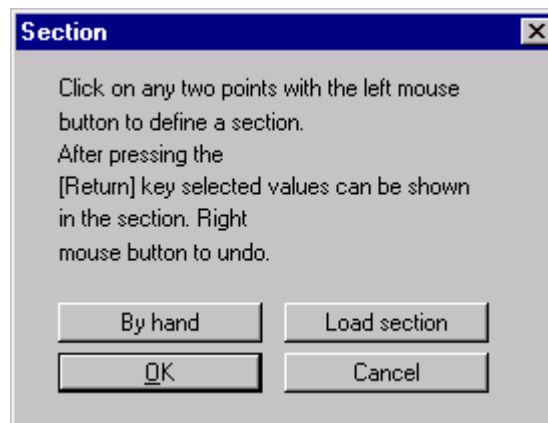
12.6 Menu option „Values in node section“

You can display state variables in a section. To do this, define a section by clicking on the appropriate nodes with the left mouse button. Intermediate points will be automatically supplemented by the program. Clicking with the right mouse button deletes the last point from the list. When the section is in accordance with your wishes, press the [Return] key. Then select the state variable to be displayed.

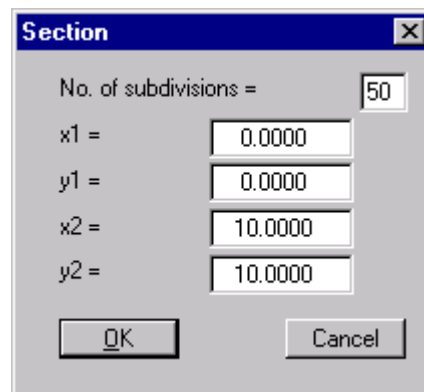
12.7 Menu option „Position“

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

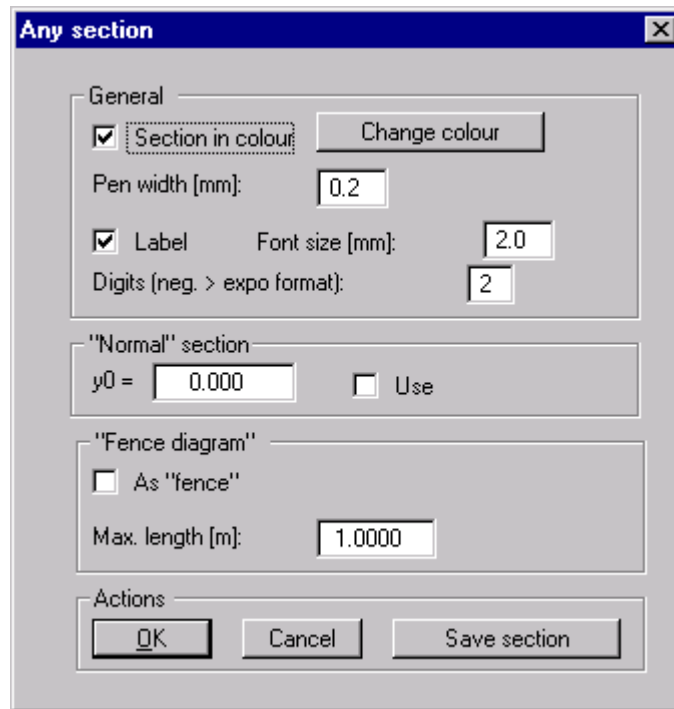
12.8 Menu option „Any section“



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. 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.



The no. of subdivisions defines at how many points section values are to be calculated. After confirming with „OK“ the following dialog box appears.



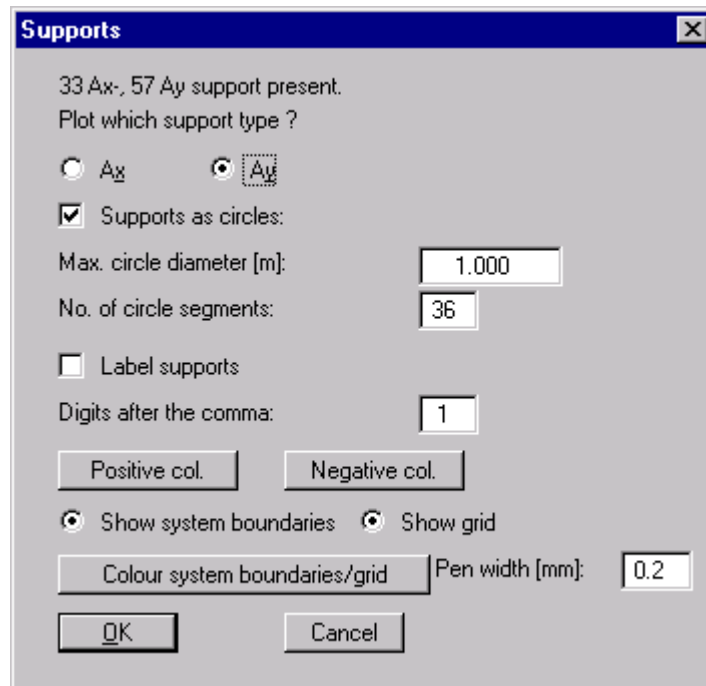
You can now decide whether the section should be coloured. By entering y_0 , you can also determine the intersection of the x with the y axis. In this case, the „Use” switch must be activated. If you do without this option, the x axis will cut the y axis at the smallest y value. You can save the section, and load it again at a later date (see above).

12.9 Menu option „Position“

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

12.10 Menu option „Support“

With this menu option you can display circle graphics of the bearing forces.



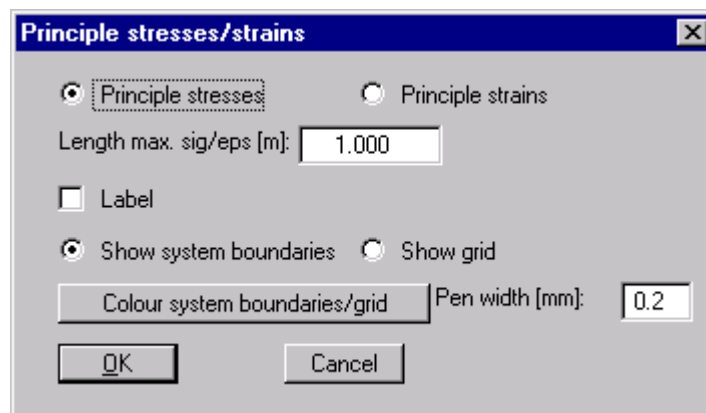
In the upper part of the dialog box you set the type of support to be determined.

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

All further input is identical to input for the menu option „Circles“.

12.11 Menu option „Principle stresses“

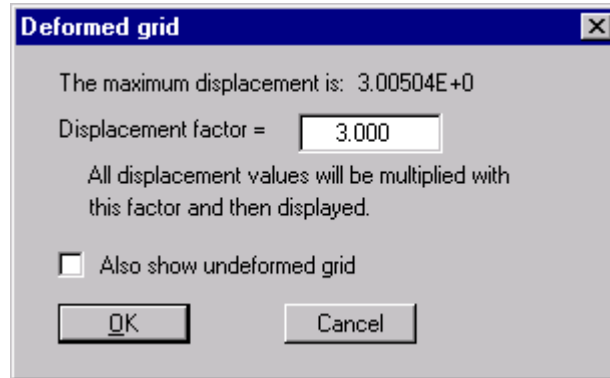
You can have the principle displayed as values-dependent lines.



The line colour and line strength can be set differently for positive and negative principal moments in the menu option „View / Pens“.

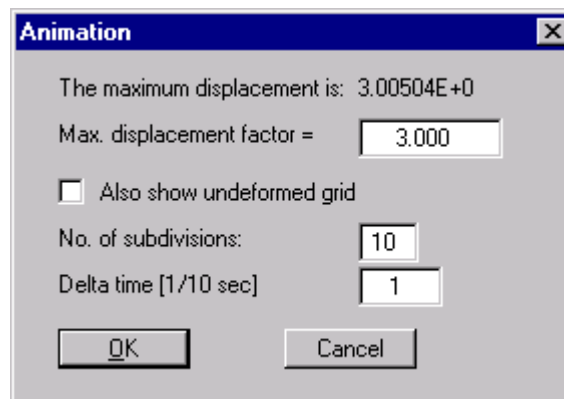
12.12 Menu option „Deformed grid“

You can have the deformed grid 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 grid can also be displayed.

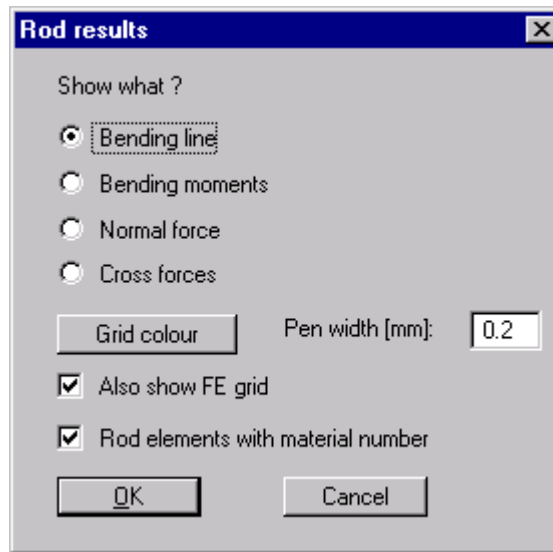
12.13 Menu option „Animation“



Similarly to the previous menu option, the system deformations can be multiplied by a factor. The presentation will be repeated in steps, the number of which you must enter. This gives an animated „cartoon“ effect. The current animation can be cancelled with the click of a mouse button.

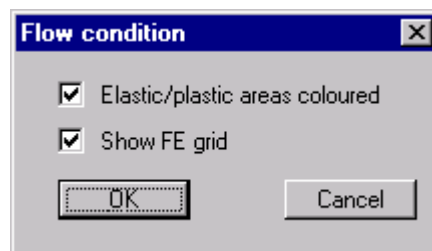
12.14 Menu option „Rods“

If rods are present in the system, you can have their state variables graphically presented.



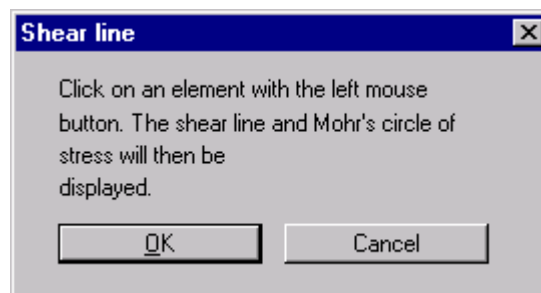
12.15 Menu option „Flow“

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 option „Pens“ with the „elastic“ and „plastic“ buttons.

12.16 Menu option „Mohr“



12.17 Menu option „Individual values”

By clicking a node with the left mouse button, you can view all data for this node.

13 Menu item ?

13.1 Menu option „Copyright”

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

13.2 Menu option „Maxima”

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

13.3 Menu option „Help”

You will see this manual as a Windows Helpfile.

14 Tips

In the menus and dialog boxes, some letters are underlined. By pressing [Alt] and the corresponding letter simultaneously, you can call up the program function via the keyboard.

Almost all function keys are assigned program functions. The allocations can be taken from the menus.

With the cursor keys and the „Page up“ and „Page down“ keys you can scroll the screen using the keyboard.

You can simplify system input if you create a Bitmap file of the system using a scanner, and then import this file using Mini-CAD. The size of the Bitmap can be scaled. This is not necessary for grid generation, however. If you cannot model the system correctly with a Bitmap you can, after completing input, give a coordinate scale correction using the menu option „Grid/Edit“ and then the „Using formula” button.

You can simplify system input if you have a DXF file of the system, and then import this file using Mini-CAD. Please read the Mini-CAD manual.

15 Example

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

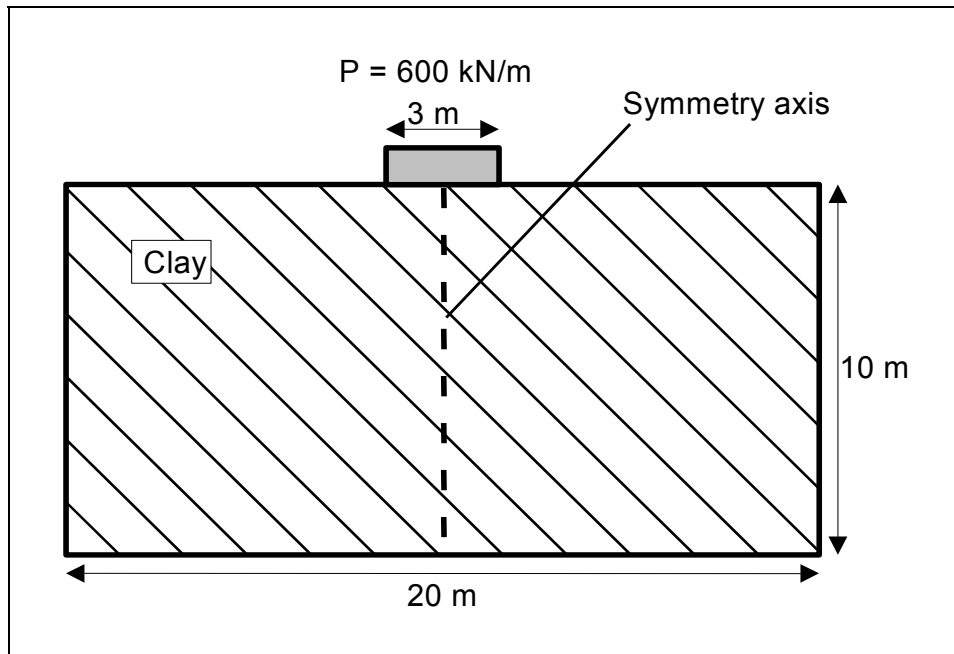


Figure 6Example system

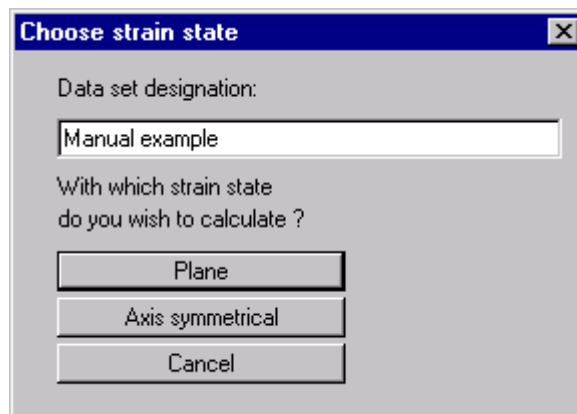
The following characteristics are given:

- constrained modulus $E_S = 7,5 \text{ MN/m}^2$
- Poisson's ratio $\nu = 0,0$
- friction angle $\varphi = 25^\circ$
- cohesion $c = 30 \text{ kN/m}^2$

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/m^2 results from the load P and the foundation width.

15.1 Step 1: Program preparation

Start the program and select the menu option „File/New”.

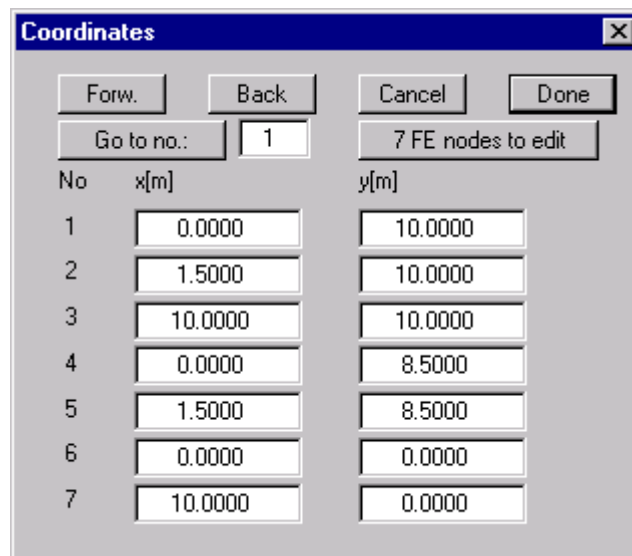


Select the „Plane” button from the dialog box.

15.2 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 option „FE grid / Edit” and then the „Using tables” button. Change the number of nodes to 7 and enter the values from the following dialog box.



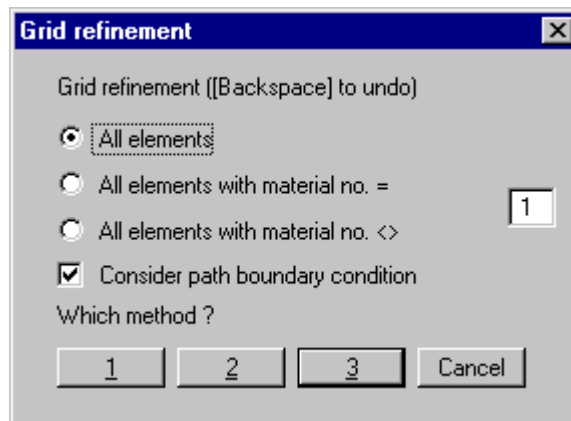
In principle, 5 points would have been sufficient to describe the system:

- four corner points
- right foundation edge

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

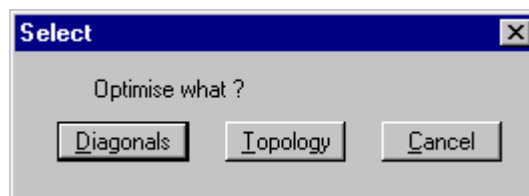
Select the menu option „FE grid / Automatic”. The program connects the seven points to a FE grid. Then go to the menu option „Page / Recompute coordinates“ to get a screen-filling system presentation.

Select the menu option „FE grid / Refine all”



and press button „3”. Repeat the grid refinement. You should now have a system with 96 triangles and 61 nodes.

Now select the menu option „FE grid / Optimise”



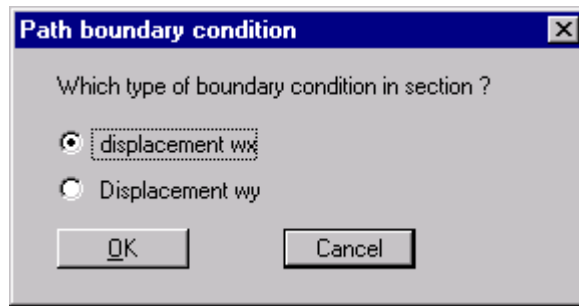
and press the „Topology“ button to get a numerically favourable grid.

15.3 Step 3: Boundary conditions

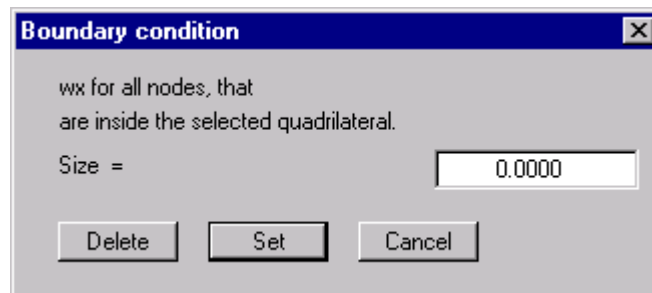
The following boundary conditions are given:

- 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 option „Boundary / Path in section”.

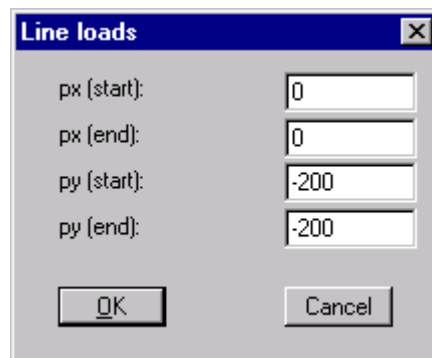


Trace the left boundary nodes and set the displacement in x-direction to „0”.



Repeat this for the lower and the right system boundaries. Select the menu option „Boundary / Path in section“ and activate the switch for path boundary conditions in y-direction. Assign the lower system boundary a displacement in y-direction of „0”.

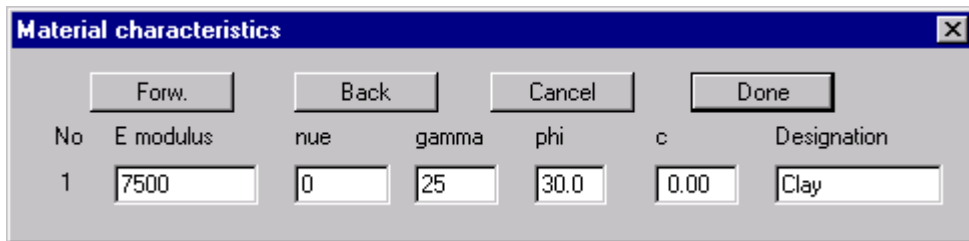
Select the menu option „Boundary / Line loads“ and click on the foundation centre and the right foundation edge. After pressing the Return key, define the foundation stress at 200 kN/m² via:



Select the menu option „FE grid / Refine all” once again, to get a finer FE grid. You should now have a system with 384 triangles and 217 nodes.

15.4 Step 4: Material characteristics

Select the menu option „System / Material characteristics”



The dialog box titled "Material characteristics" contains four buttons at the top: "Forw.", "Back", "Cancel", and "Done". Below the buttons is a table with the following data:

No	E modulus	nue	gamma	phi	c	Designation
1	7500	0	25	30.0	0.00	Clay

Enter the values into the dialog box.

Note:

If several materials were defined during definition of the triangular grid, the above dialog box will contain the correct number of rows for the individual materials.

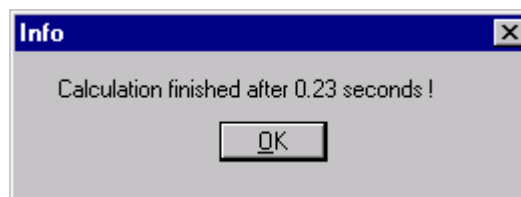
15.5 Step 5: Calculate slab

Select the menu option „System / Calculate”.



The dialog box titled "Elastik" contains the text "Calculation of plane strain state" and two buttons: "Start" and "Cancel".

Press the „Start” button. Depending upon the power of your computer, information will be shown after a certain time on the time taken to complete calculations.



The dialog box titled "Info" contains the text "Calculation finished after 0.23 seconds !" and an "OK" button.

15.6 Step 6: Evaluate

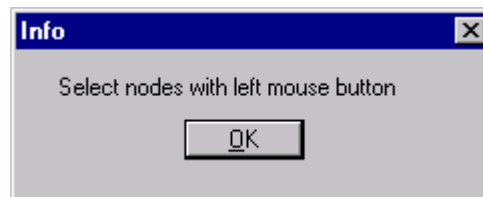
First, the results are to be compared to exact solutions. Using the relationships from the „Grundbautaschenbuch (Volume 1; „Stress determination“)" the exact values for this case can be calculated.

x [m]	Result GGU- ELASTIC	exact	Differenc e [cm]	Differenc e [%]
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

Table 1 Settlements [cm]

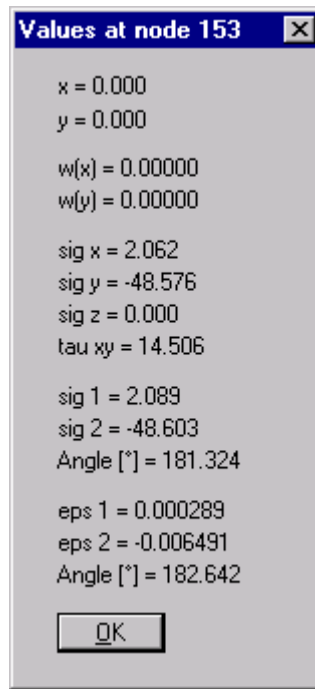
The results deviate by a maximum of 3.3 % from the exact solution.

If you are interested, as in this case, only in individual values at certain nodes, you can do simply by using the menu option „Evaluation / Individual values“.



Then click on the nodes with the left mouse button.

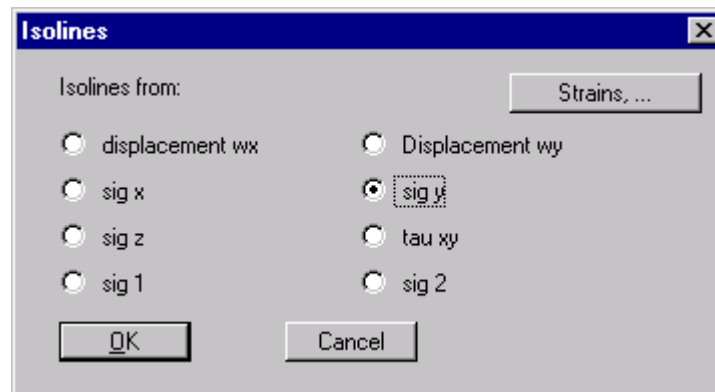
In the following dialog box the result is shown for line 1 of Table 2, Settlements (foundation centre at x = 0,0 m).



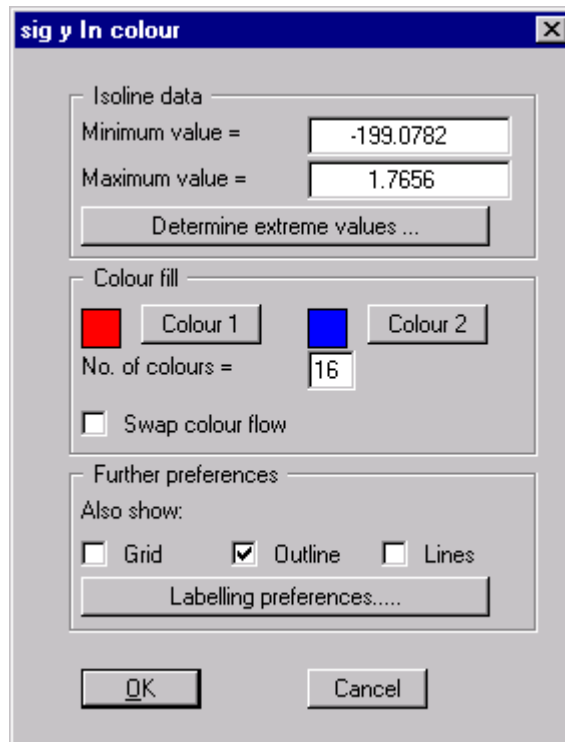
15.7 Step 7: Drawing the stress bulb

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

For this, select the menu option „Evaluation / Coloured isolines“.



Activate the „sig y“ switch and confirm with „OK“.



Press the „Determine extreme values“ button once and then the „OK“ button. Voila ! the stress bulb is in colour. If you would like to print the stress bulb, go to the menu option „File / Start“. In a similar manner you can create isolines of displacements, strains, etc. (menu option „Evaluation / Normal isolines“ etc.), or determine bearing forces (menu option „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 Mini-CAD, create a page header, or load the supplied header (ELASTIK.KPF) using CAD for header data. Please read the Mini-CAD manual.