BE-SLABS

Manual

June 2017

Friedel Hartmann

www.be-statik.de
www.winfem.de

hartmann@be-statik.de

1.0

1. INSTALLATION

1.1. Starting the program

To start click on the file

BE-PLATES.EXE

in the program directory or on the corresponding icon on the desktop.

1.2. The main menu

To input a plate, click on

Dimensions // shape of the plate

next on

LC's // loads

then on

Points // stress points (like mesh points in FE-analysis)

and finally start the calculation by clicking on

Calc **//** analysis of the plate

The stress points are the points where the program outputs the bending moments, shear forces, etc.

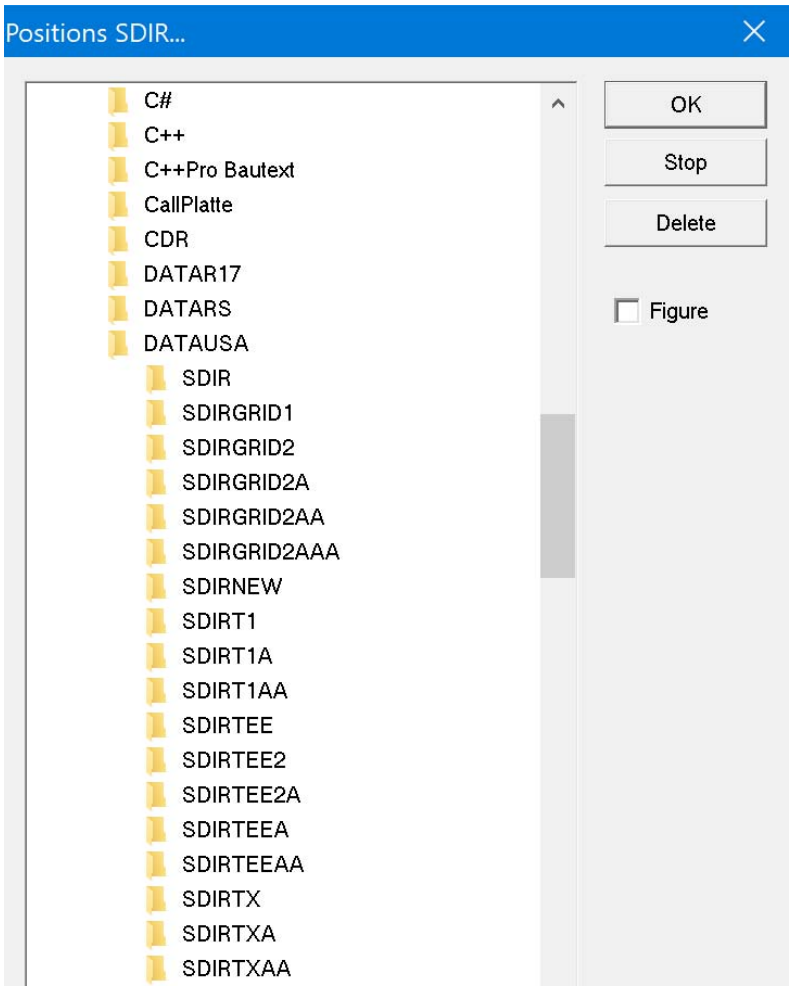
1.3. Positions

To each plate is assigned a position number such as

123 200 3400 Cellar 234Floor rooftop etc.

Blanks, as in **A 23**, are allowed but special characters as in **231.2** or **A+1** are not.

The files that belong to a position XYZ are stored in the folder (subdirectory) **SDIRXYZ.** A click on the button **Open** in the main menu will show you a list of the subdirectories stored in the current data path.

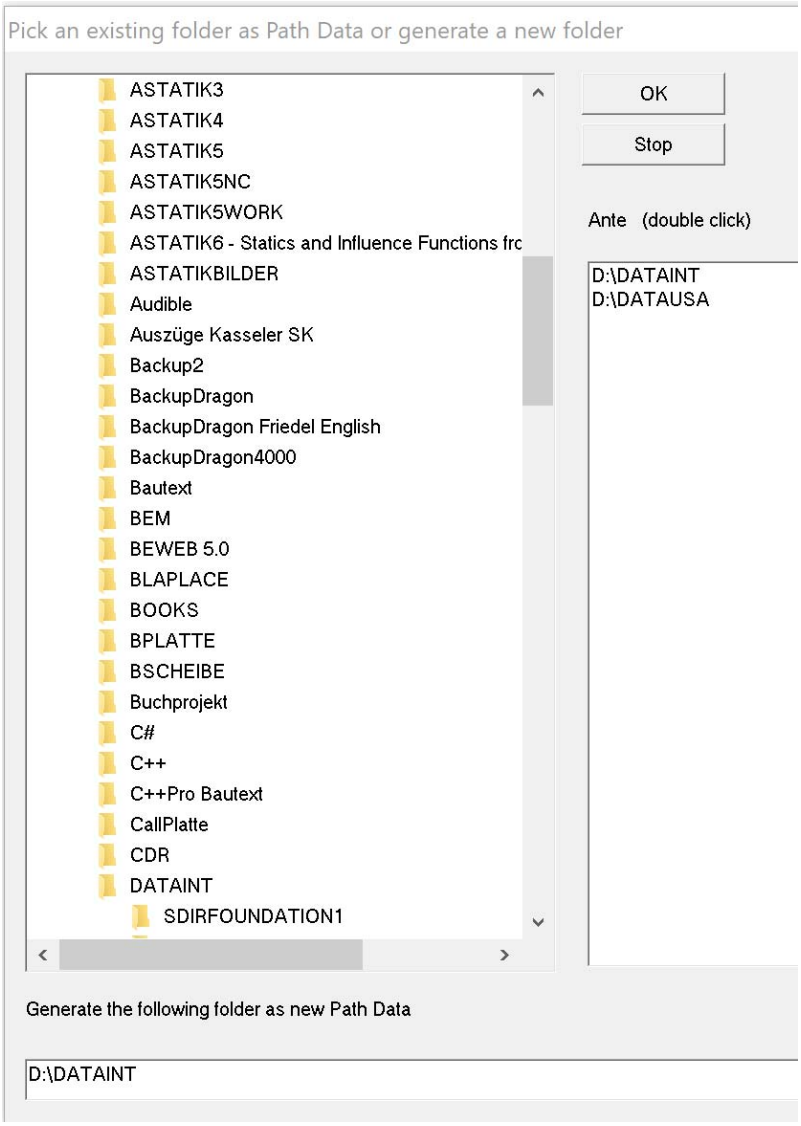


1.4. Path Data

The data which belong to position 100 are stored in the folder SDIR100 and the data which belong to position 200 are stored in the folder SDIR200. The root directory for these folders is what *path data* points to. It can be any directory on the hard drive or on a network.

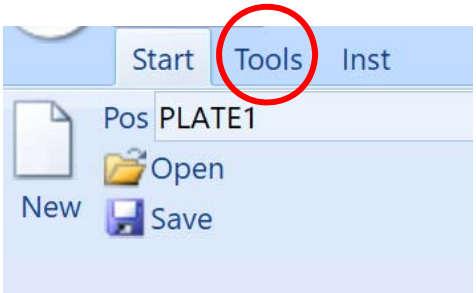
To change the path, click on the button

in the main menu.

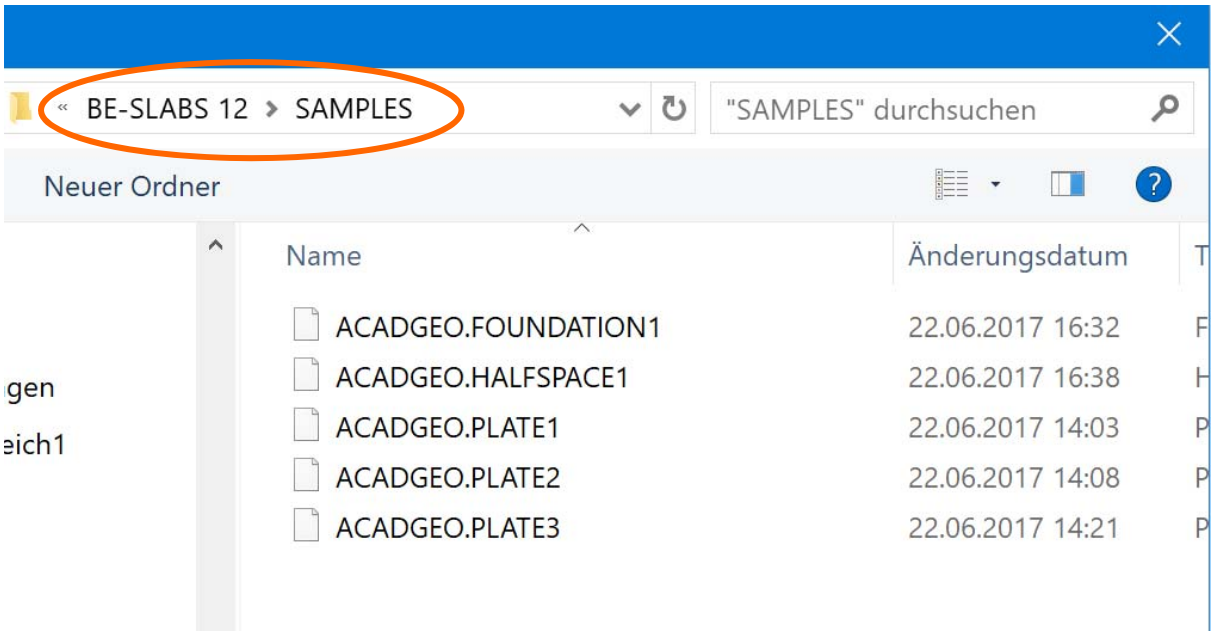


1.5. First steps

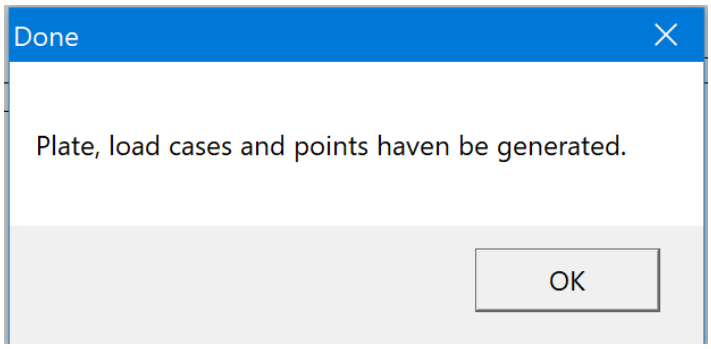
1. Click on the button Tools



2. Navigate to the directory where the program is stored and find the folder *Samples.*



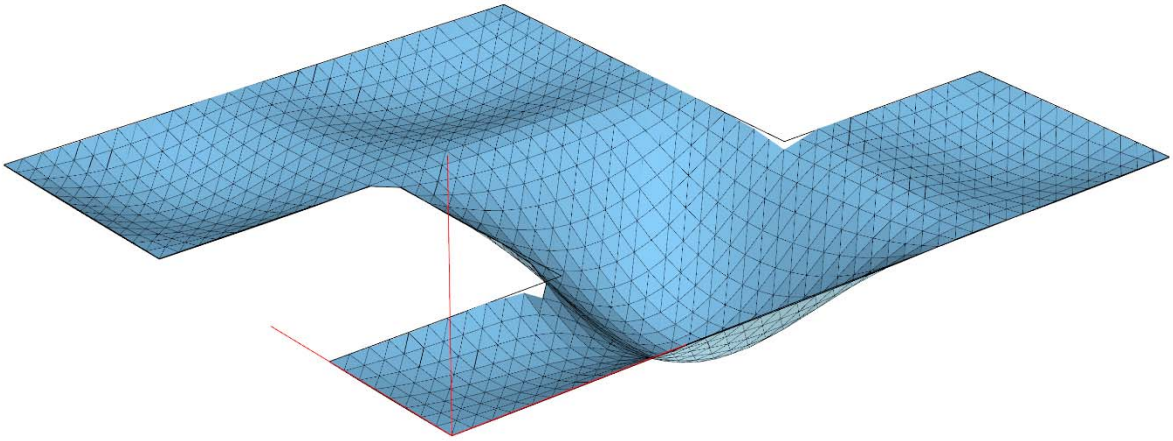
3. Double click on the file ACADGEO.PLATE1 and the position PLATE1 will automatically installed



4. In the main menu click on the button

to start the analysis and when the program is finished click on the button

To display the results.



Some useful shortcuts in graphics mode:

Turn the mouse wheel to scale figures.

The up and down keys let you switch between load cases.

Zoom: click the left mouse button and open a box (window)
Zoom undo: press ESC-key

Ctrl + Mouse wheel = Font size
Shift + Mouse wheel = Size of markers for stress points
Alt + Mouse wheel = Line width

In some windows text (annotations) can be relocated by clicking on the text and pushing the text with the mouse in any direction.

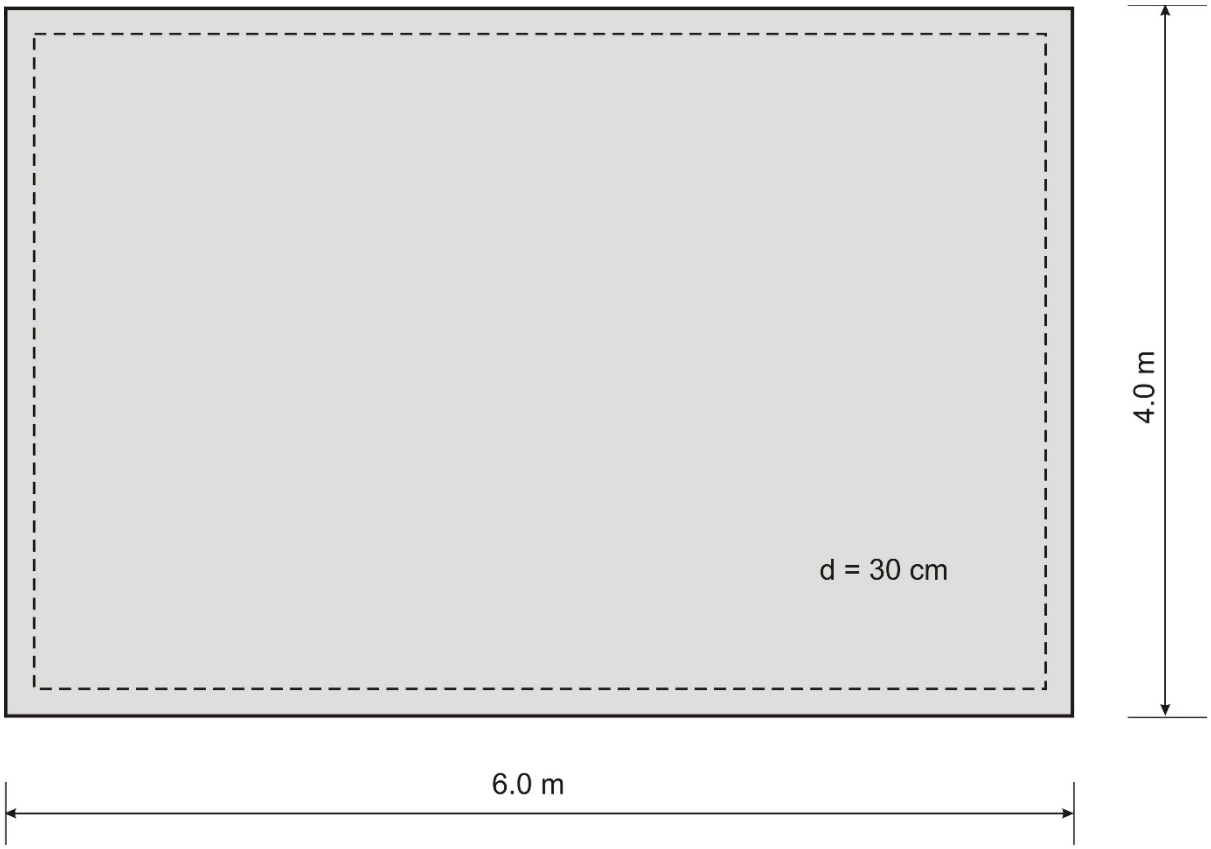
In some windows, the font size can be changed 'on the fly' by pressing the shift-key and turning the mouse wheel.

2. INTRODUCTION

A simple hinged plate

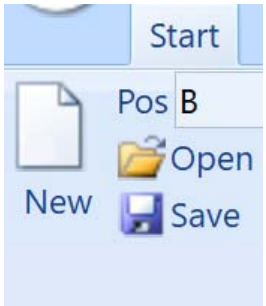
size: 6 x 4 m
thickness: 30 cm
concrete: C 20 (E = 3.0 10^7 kN/m^2)
Poisson's ratio = 0.16
hinged support on concrete walls
gravity load + partial area loads + ...

serves as introductory example.

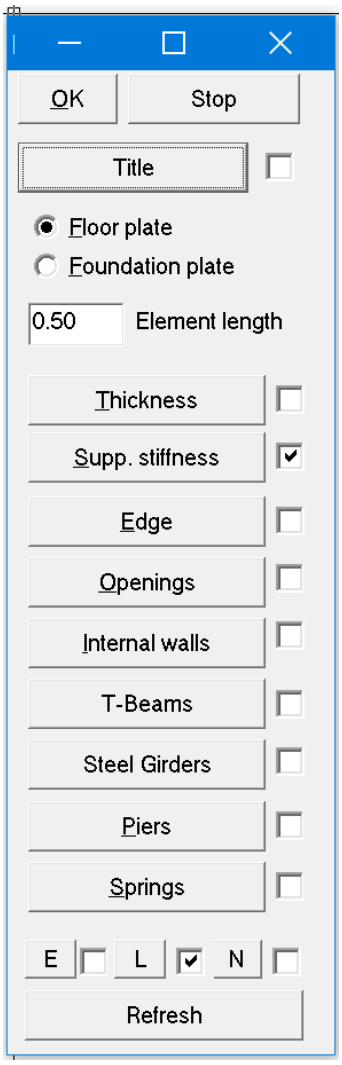


2.1. Entering the plate

Click on New and enter the position number P1

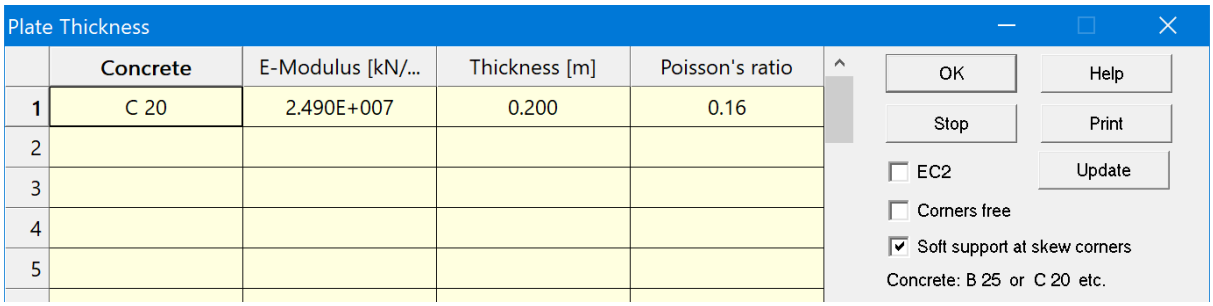


To input the plate, we use the *dialog input mode*, that is we work our way through the following dialog:



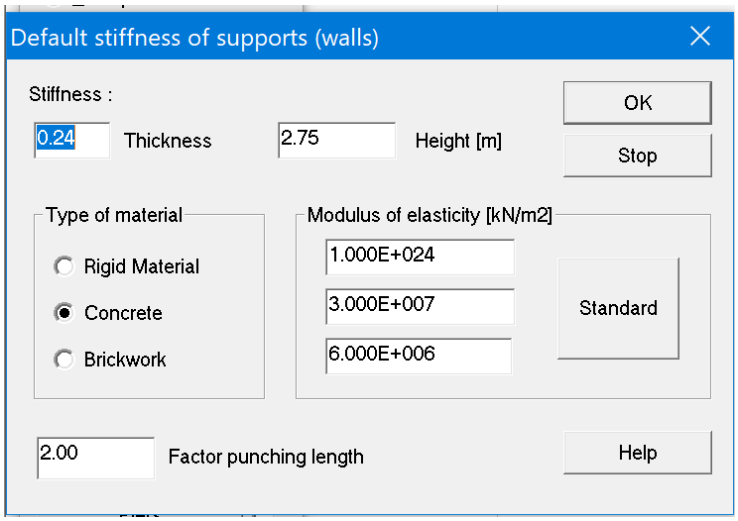
If the dialog is not visible you can activate it by clicking on the entry **Dialog input.**

Next specify the thickness of the plate and choose the type of concrete (or any other material)



The values of the modulus of elasticity are preset for C 20, C25 etc. If you enter a 0 in the first column you can specify any value of the modulus of elasticity.

Next on the main dialog click on Supp. stiffness and enter the specific values for the supports.



At the ends of internal walls, the support reactions tend to increase - sometimes dramatically. At such ends the program calculates punching forces by integrating the support reactions over a length of 2.0 times the thickness of the walls. This 2.0 (or any other value) is the factor for the **punching length**.

Next on the main dialog click on the button Edge and enter the coordinates of the four vertices of the plate.

Edges of the plate

	x [m]	y [m]	Support	Stiffness	Rot. Stiffness	Ten.
1	0	0	H	2.618E+006		Y
2	6	0	H	2.618E+006		Y
3	6	4	H	2.618E+006		Y
4	0	4	H	2.618E+006		Y
5						

The four sides of the plate are hinged (H). The numerical values for the vertical stiffness per unit length

2.618E+006 = E A /l = E * (1.0 * h) /l = 3.00E++007 * (1.0 * 0.24) / 2.75 kN/m

of the four sides of the edge is based on the values specified in the previous dialog '*Stiffness of supporting walls*'.

The support stiffness of each of the four sides can be edited.

If the default values for the stiffness of the supports (previous dialog) have been modified *after* you worked in this dialog and you want to change the default values in this dialog accordingly then click on the button Update stiffness.

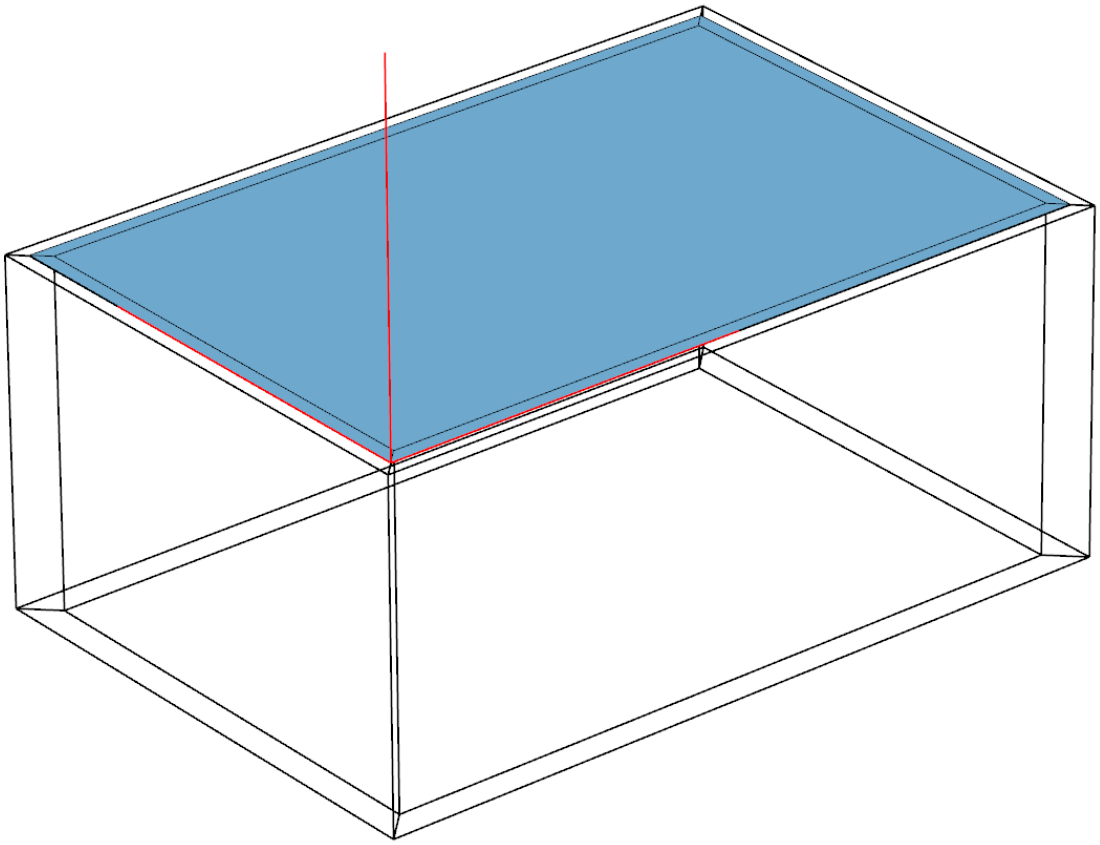
After the input of the four corner points, click on OK, and call the discretization of the plate by clicking on the icon

Shortly after you'll see a log file with an echo of the input.

D:\DATAINT\SDIRP1\D1.TXT

```
Plate analysis with boundary elements   BE-SLABS Version 12.2
-------------------------------------------------------------
Position P1
-------------------------------------------------------------
Date: Monday  26. 6. 2017  D:\DATAINT\SDIRP1
---------------------------------------------

Material

Thickness h =  20.0 cm   I = 1.0 * h^3/12 = 6.67E-004 m4
Concrete C 20/25 E =  24900.0 MN/m2  (DIN 1045-1)
EI              =  16.60 MN m2
Poisson's ratio =  0.16

Edge supports consist of: Concrete
E-Modulus = 3.000E+007 kN/m2  height = 2.75 m  thickness = 0.24 m

Stiffness = E * A / height = 3.000E+007 * 0.24 * 1.0 / 2.75 = 2.618E+006 kN/m

Average element length = 0.50 m

Dimensions

Edge
----
Sides                Length    Support      Vert.       Rot.
                               condition  stiffness   stiffness
Nr.  x [m]  y [m]     [m]                  [kN/m2]     [kNm/m]      Ten.   shape
 1   0.00   0.00     6.00     hinged       2.62E+006   0.00E+000     Y      S
 2   6.00   0.00     4.00     hinged       2.62E+006   0.00E+000     Y      S
 3   6.00   4.00     6.00     hinged       2.62E+006   0.00E+000     Y      S
 4   0.00   4.00     4.00     hinged       2.62E+006   0.00E+000     Y      S
```

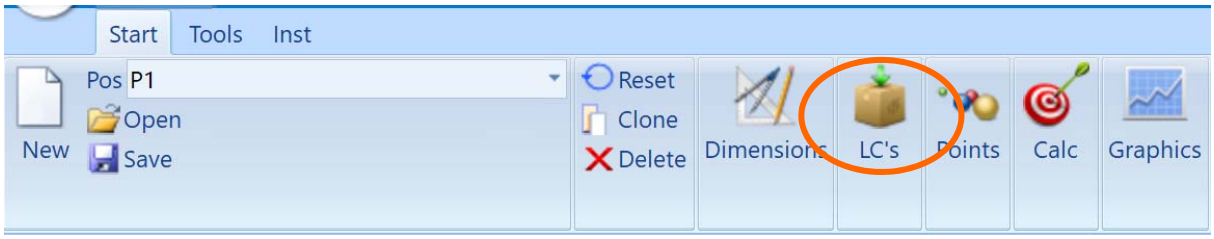
and a 3-D figure of the plate



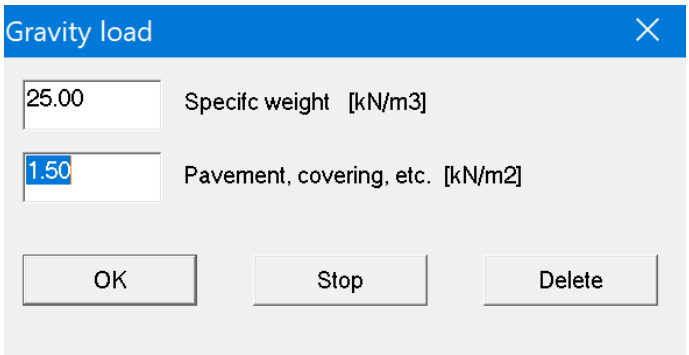
This concludes the input of the plate and we return to the main menu of the program.

2.2. Load cases

In the main menu click on the entry **LCs** to specify the loading.

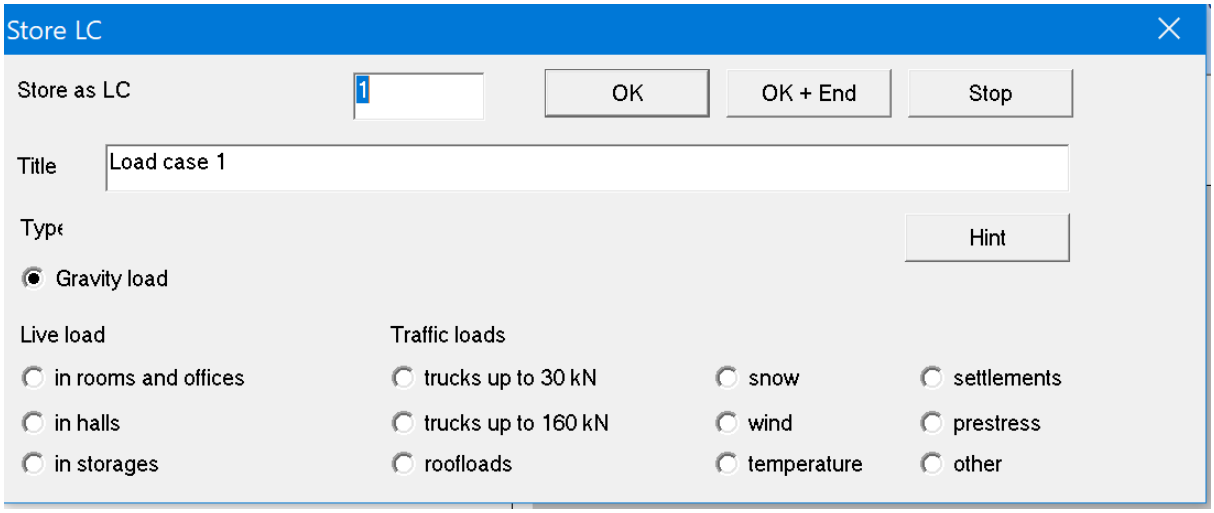


Choose Uniform load -> Gravity load



Save the load case by clicking on the icon

Confirm the default entries



and click on OK (not on OK + End) because other load cases will follow – we are not finished yet.

Load case # 2 is a partial area load which – for simplicity – covers the whole plate, that is the partial area is the whole plate.
To clear the plate, click on the icon

to prepare for a new input. To start the input of the partial area, click on the icon

and next on the four vertices of the plate: Start at the lower left corner and then traverse the plate in counter clockwise direction till you are back at the starting point. You could as well hit the C key to close the polygon if you are at vertex # 3.

Enter a value of 5 kN/m² for the magnitude of the uniform load. Finally click on the icon

to save the input as load case # 2.

Clear the plate to prepare for the next load case. In the third load case, the partial area load will only extend over the left part of the plate.

Click on the icon

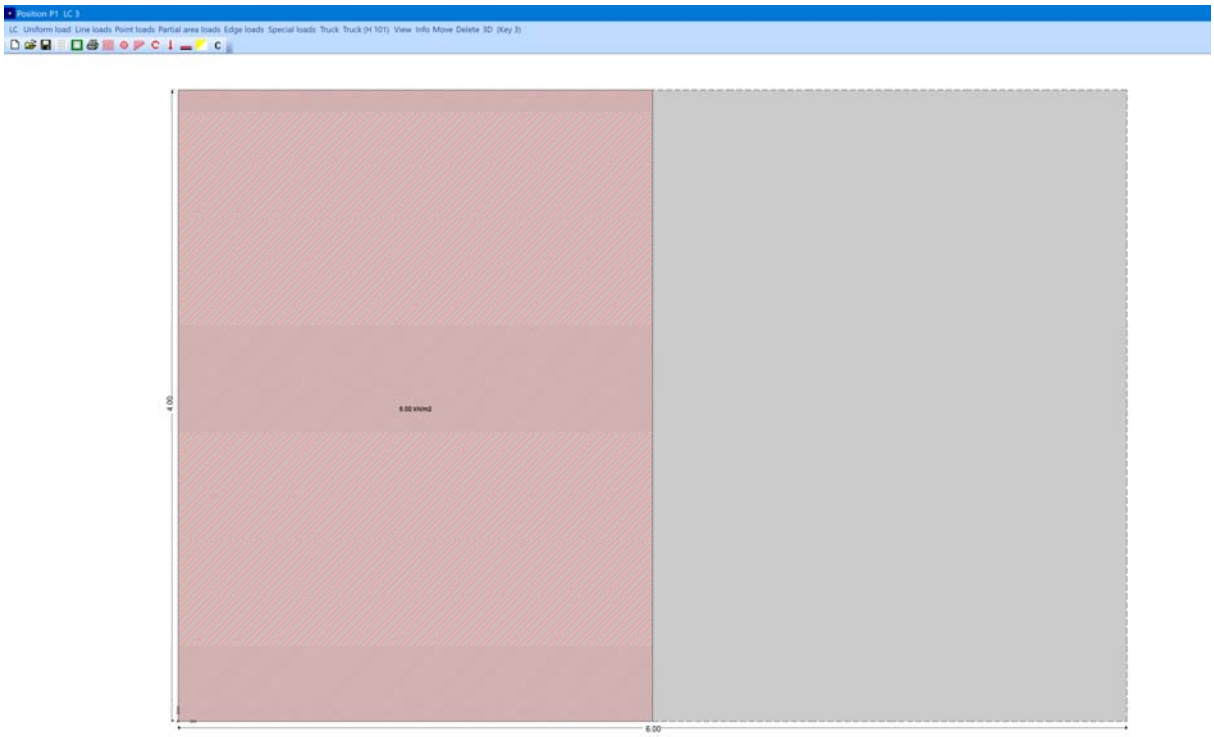
And click on the lower left corner, move the mouse to the upper right corner, and click there as well. Press the right mouse button and specify the value of the partial area load.

A click on the menu entry Partial are load will display the entries of the loaded area

Shape (vertices) of loaded areas

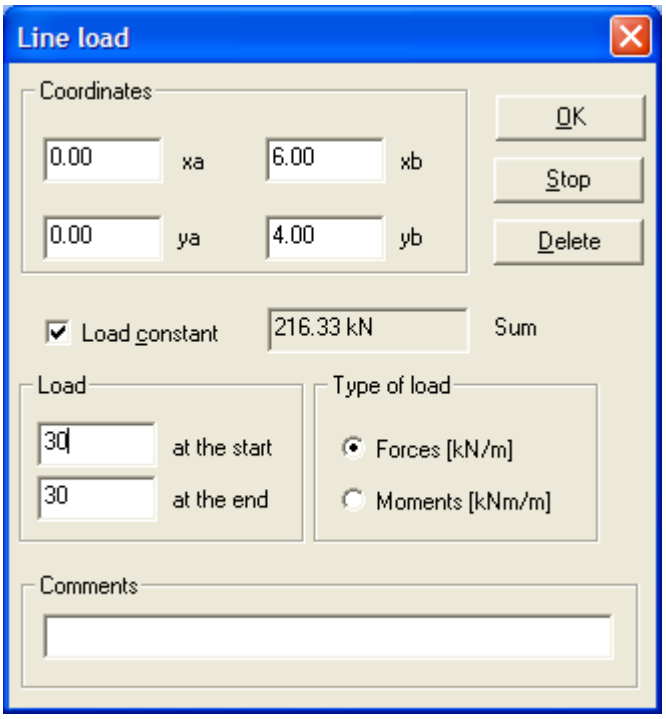
	x [m]	y [m]	p [kN/m2]
1	0.00	0.00	5.00
2	3.00	0.00	5.00
3	3.00	4.00	5.00
4	0.00	4.00	5.00

You could use this table-input mode as well to specify other partial area loads.



In load case # 4 a line load is acting along the first diagonal of the plate. To start click on the icon for line loads

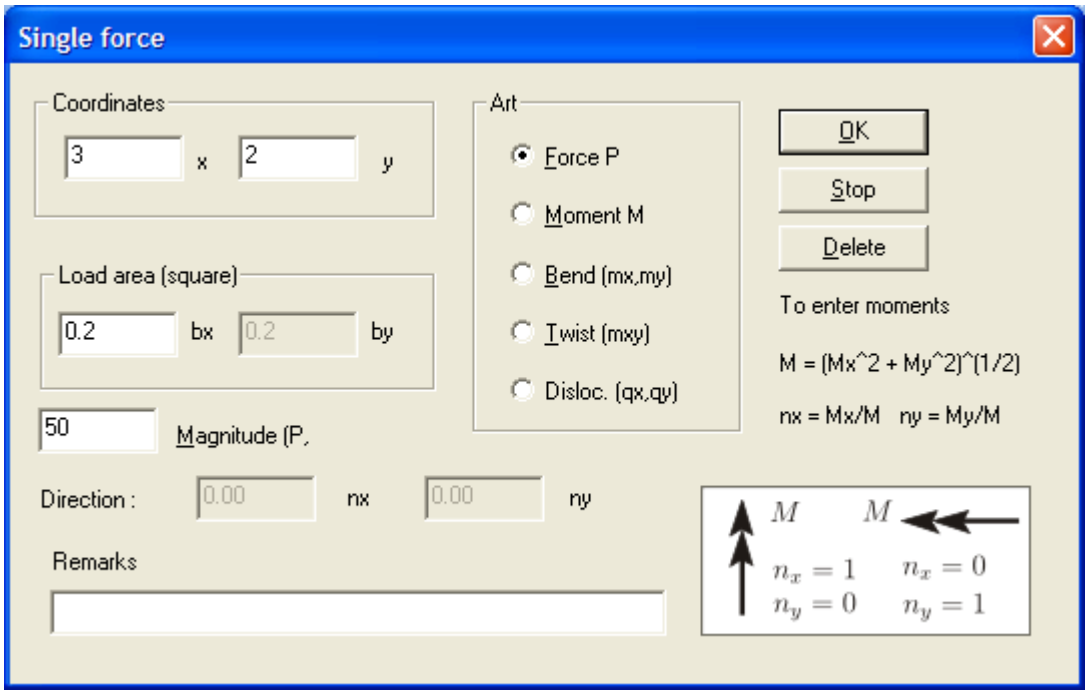
Click on the lower left corner, draw a line to the upper right corner and click again. Enter a value of 30 kN/m for the line load



Save the load case, clear the plate and prepare for the input of the next load case.

In load case # 5 we place a single point force of 50 kN at the center of the plate. Click on the icon

and next click at the (approximate) center of the plate. A dialog opens where you can specify more precisely the exact coordinates of the point force and the magnitude of the force:



Store this last load case and return to the main menu of the program.

2.3. Input of stress points

The stress points are the nodes, the Gauss points, so to speak, in a BE-program. At these points are calculated the bending moments, the shear forces, and the deflection. Because there is no mesh these points must be generated separately in a BE-program. Because the accuracy does not depend on the mesh size the grids or meshes can have any shape and any size.

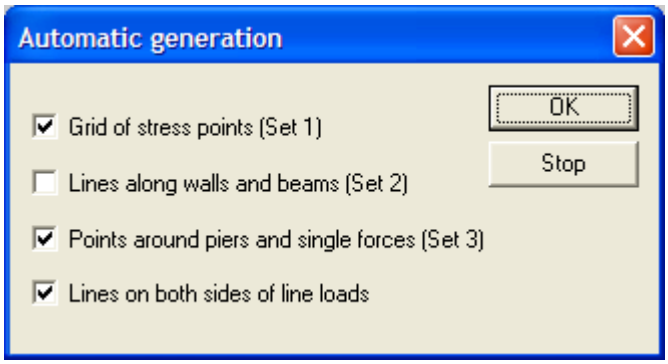
Stress points come in different sets.

#1: The first set is usually a regular grid of stress points which covers the whole plate.

#2: Set number two consists of a set of single lines of stress points which run along the axes of walls and T-beams. They serve to calculate the bending moments across these supports.

#3: Set number three consists of clusters of points (four points each) which are centered at the piers and they serve to better catch the steep gradients of the bending moments near the piers and near point forces.

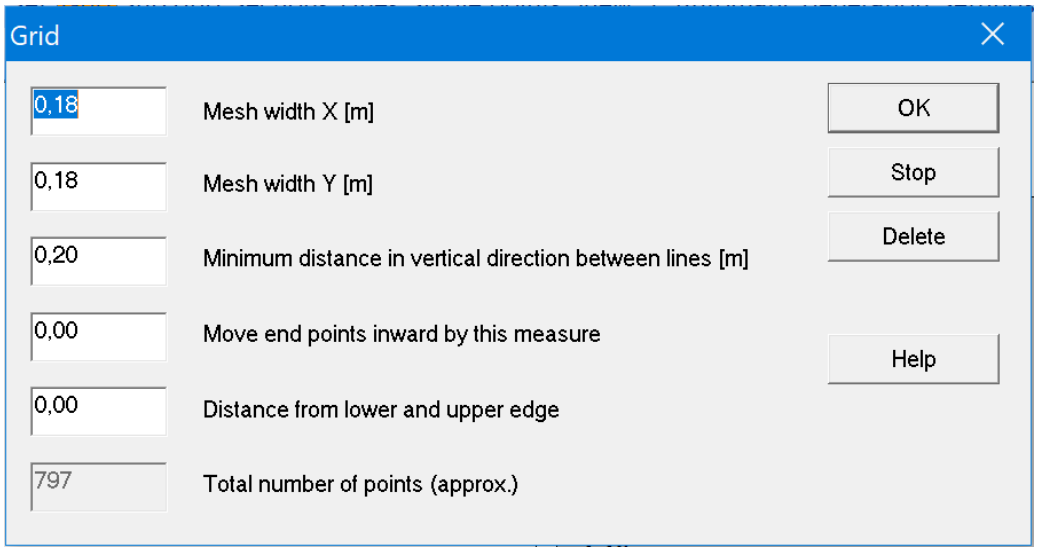
At the start of the program you can choose these three default sets



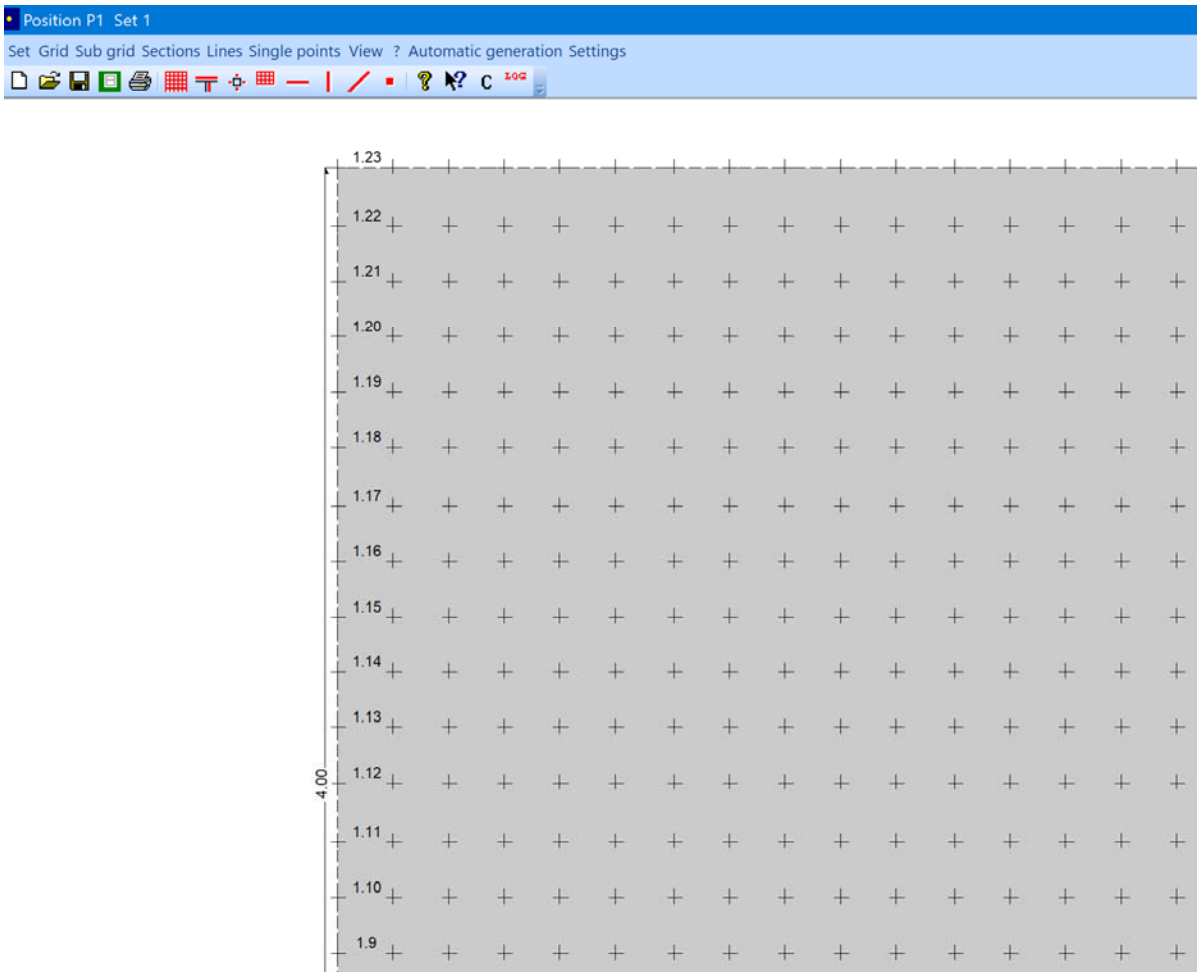
but you can also generate your own sets.

The grid (mesh) is determined by the mesh width in horizontal and vertical direction. The grid is not a mesh but rather a set of horizontal lines on which the points are distributed in (approximately) equal distance. The distance in vertical direction is controlled by the mesh width in vertical direction.

The program will try to align the mesh lines near openings in such a way that the lines will touch the lower or upper edge of an opening. If the edges of two openings – sitting side by side - are almost on the same level this would force the mesh to have two horizontal stress lines with nearly zero vertical distance. To avoid this, you can assign a minimum distance in vertical direction to the stress lines.



Unlike FE-methods the accuracy does not depend on the mesh size. And each point is independent in the sense that the stresses and deformations are calculated by influence functions which (mainly) live on the boundary and not by interpolating functions in between the nodes.

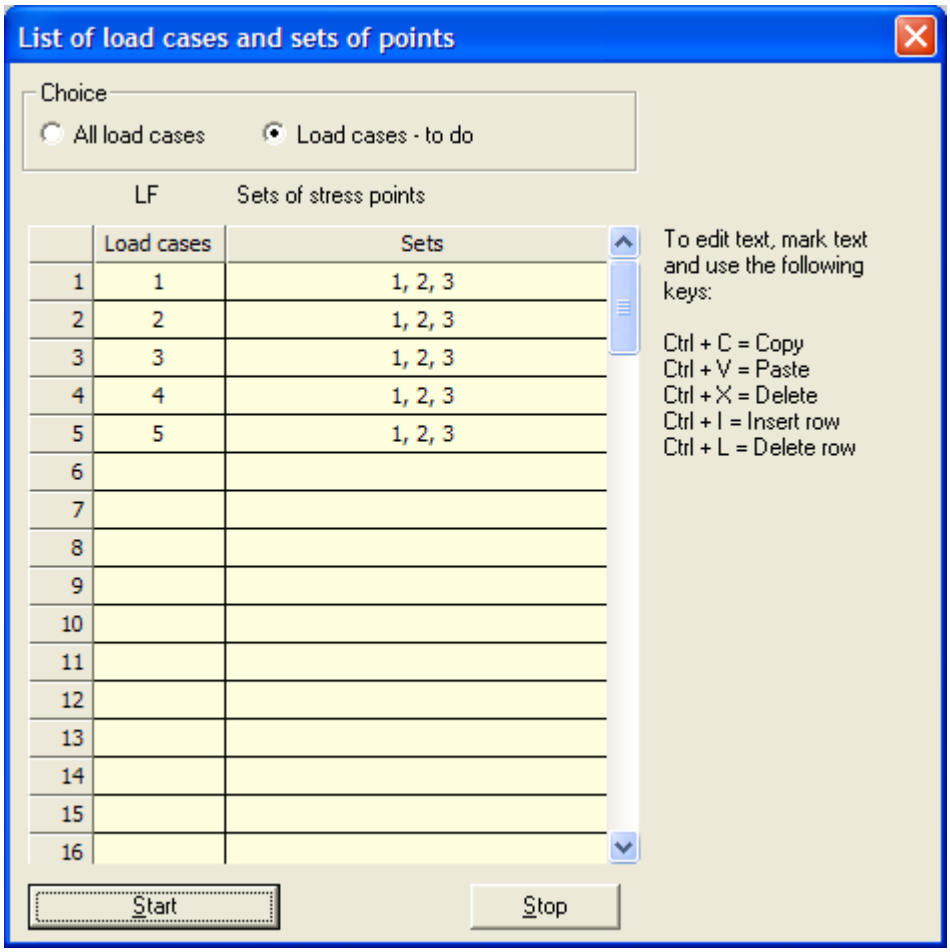


2.4. Start

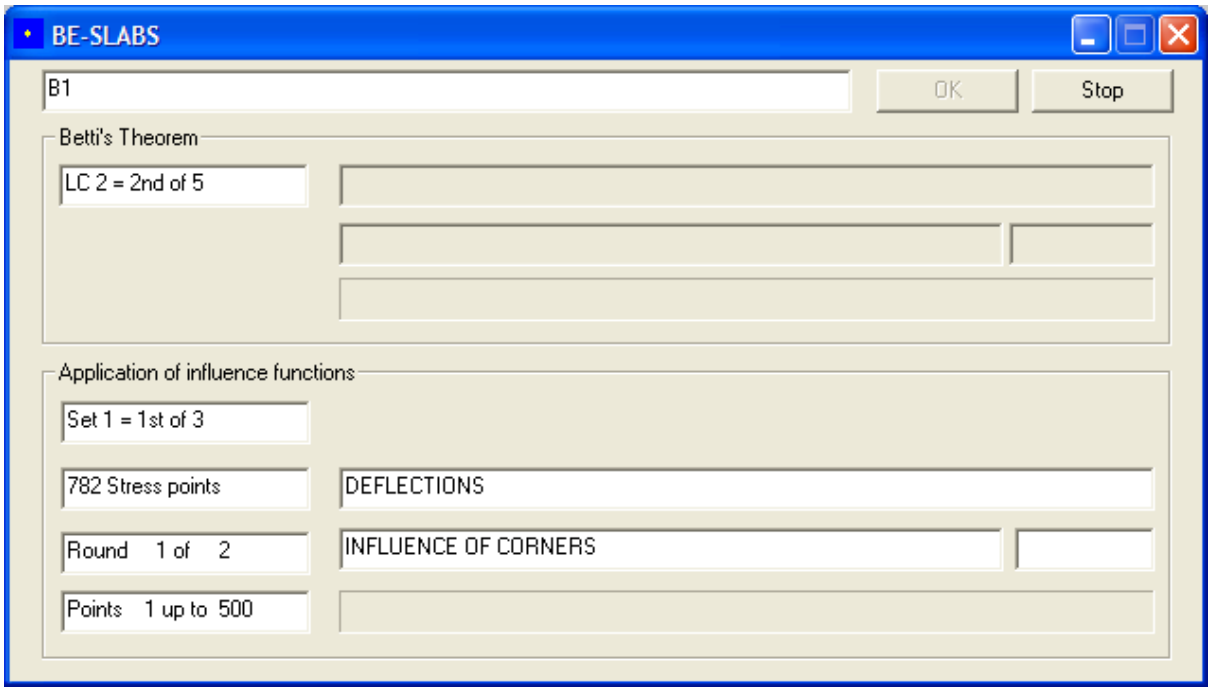
To start the analysis (getting results) in the main menu click on

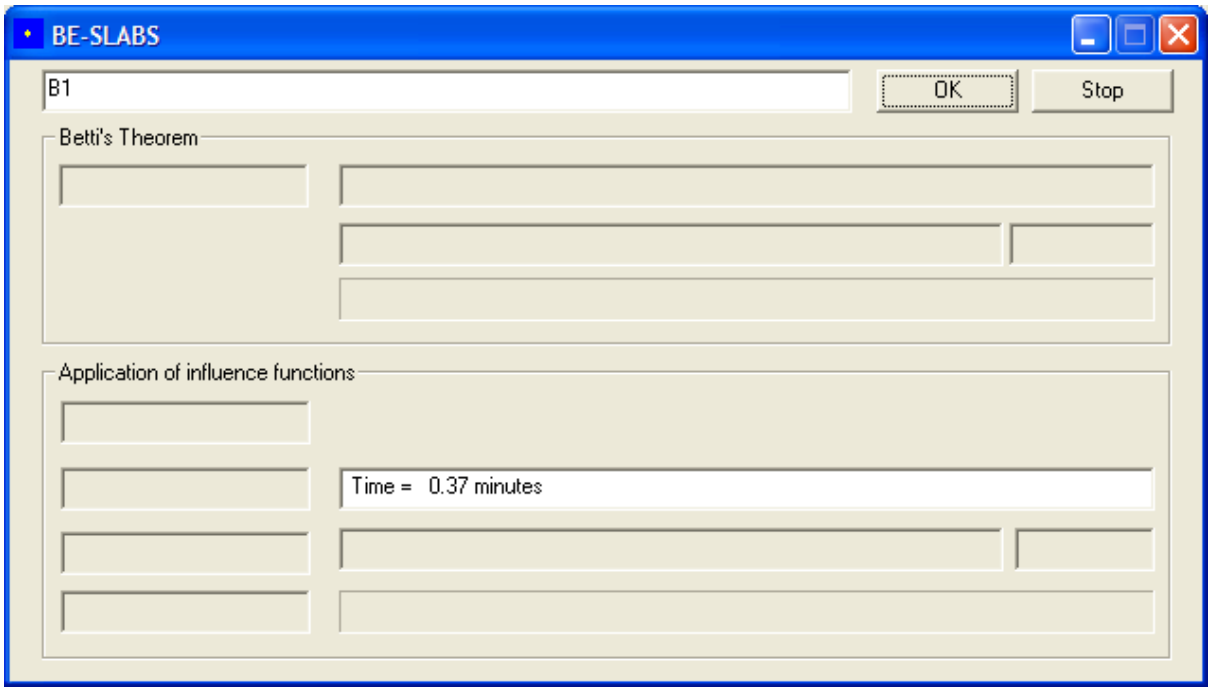
Calc

and in the following dialog on the button Start.



You can follow the action on the screen.

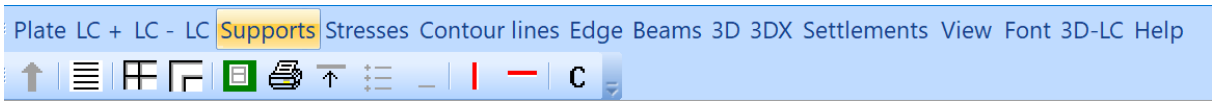




2.5. Graphics

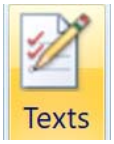
To display the results, click on the button Graphics in the main menu

and pick any of the following options



2.6. Reading and printing text files

To read and to print the text files with the results of the analysis click on the icon



in the main menu and double-click on the corresponding file.

2.7. Additional examples

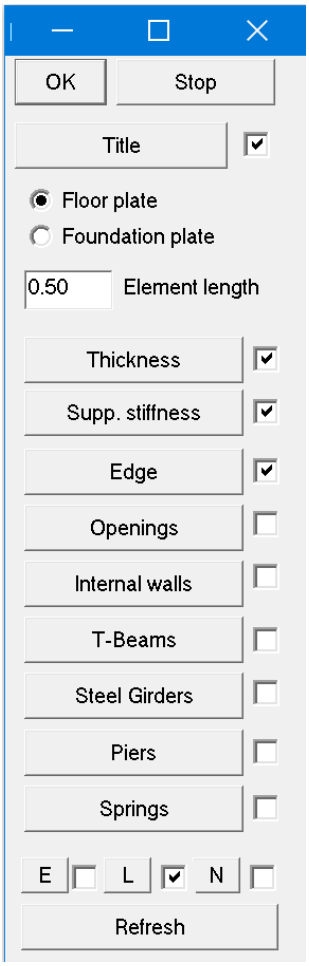
In the folder *Samples,* you can find additional preset problems which you can load and study.

3. Plate

The shape of the plate can either be entered with the keyboard, can be retrieved from a DXF file, or you can use the mouse to draw the plate on the screen (graphical input mode). To start the input, click on the button Dimensions in the program menu.

3.1. Dialog input mode

Dialog input mode is simple and easy to use. It is the default mode. If the dialog is not displayed



click on the corresponding entry Dialog input in the menu bar.

In this mode, you use standard dialogs and tables to detail the shape of the plate. You start at the top of the main dialog and you then work your way through the different dialogs.

If you mix graphical input mode with dialog input mode then, to update the entries in the dialogs at the end of the graphical input mode, click on the button Refresh at the bottom of the input dialog.

When you are done with the input of the plate, click on the icon

to call the discretization of the plate where the edge(s) of the plate is partitioned into boundary elements and the plate is prepared for the numerical analysis.

3.2. Graphical input mode

In graphical input mode, you first create a line drawing of the plate and in the second step you assign a 'meaning' to the single lines. Graphical input mode is only recommended for advanced users.

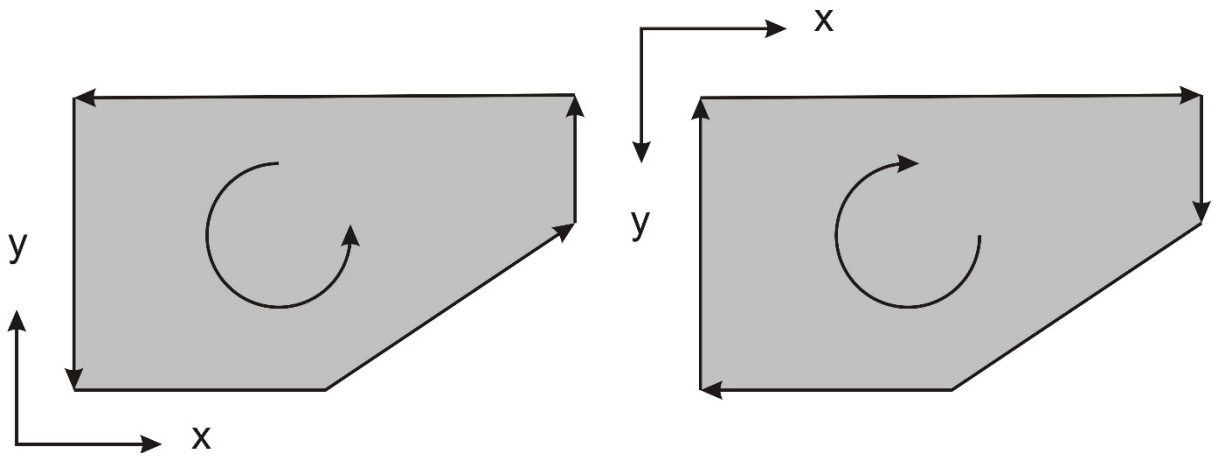


Fig. 3.1 The two possible systems of coordinates. The program uses the left system. When you this system the plate always lies to the right when you move on the edge in counterclockwise direction.

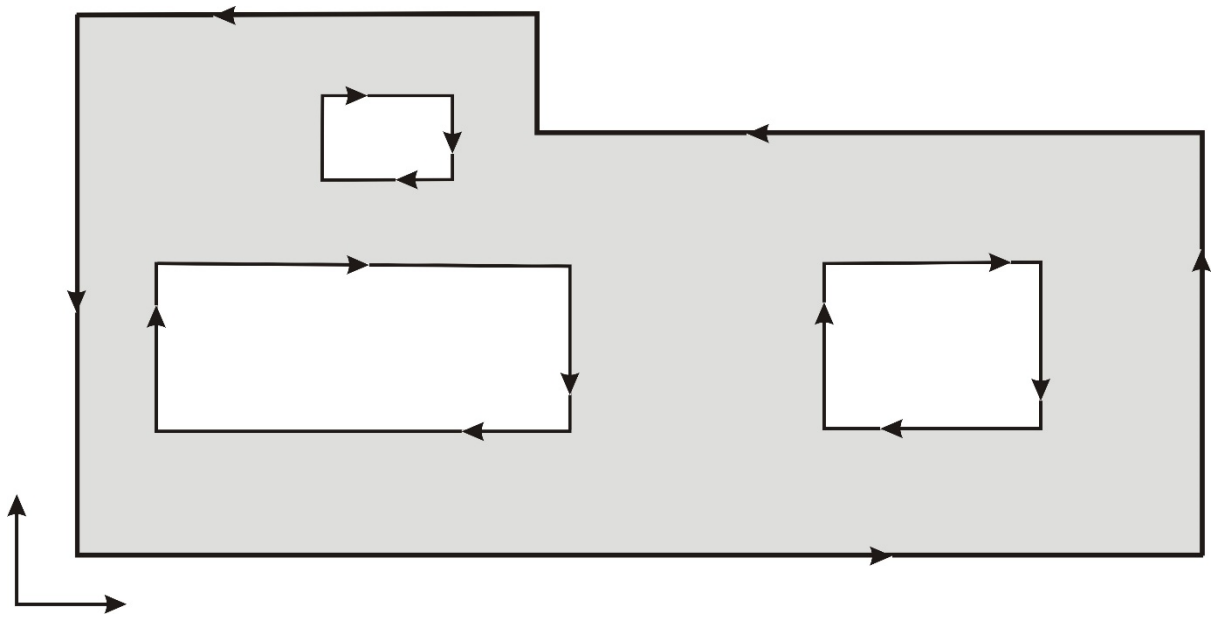


Fig. 3.2 Along the edges of openings the sense of rotation is opposite to the outer edge.

To start click on the icon of the tool you want to use

 a polygon  a single line

To close a polygon, press the C-key or click with the mouse on the starting point.

At each point, you can save your input by clicking on the icon

The input ends with the **discretization** of the plate, that is, the subdivision of the edge of the plate into boundary elements. To do this click on the icon

The subdivision of the edges and walls into boundary elements, etc. is done by the program.

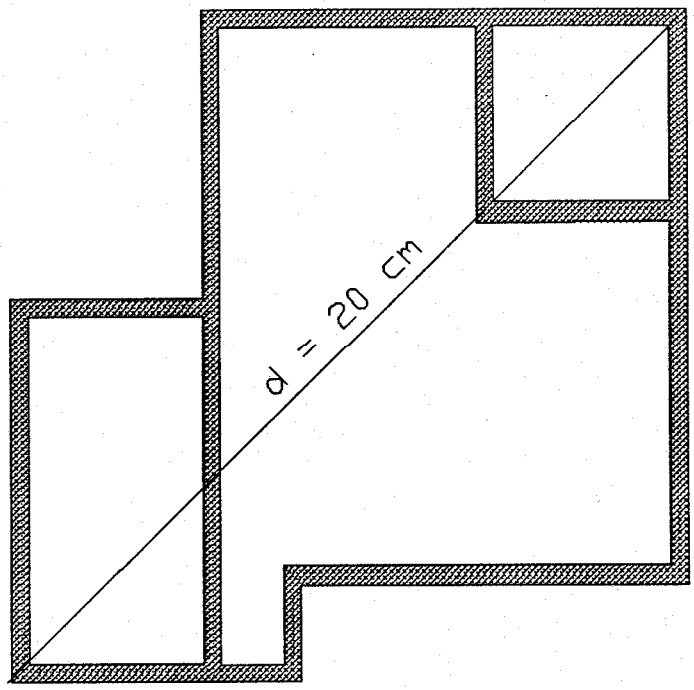


Fig. 3.3 A plate with uniform thickness

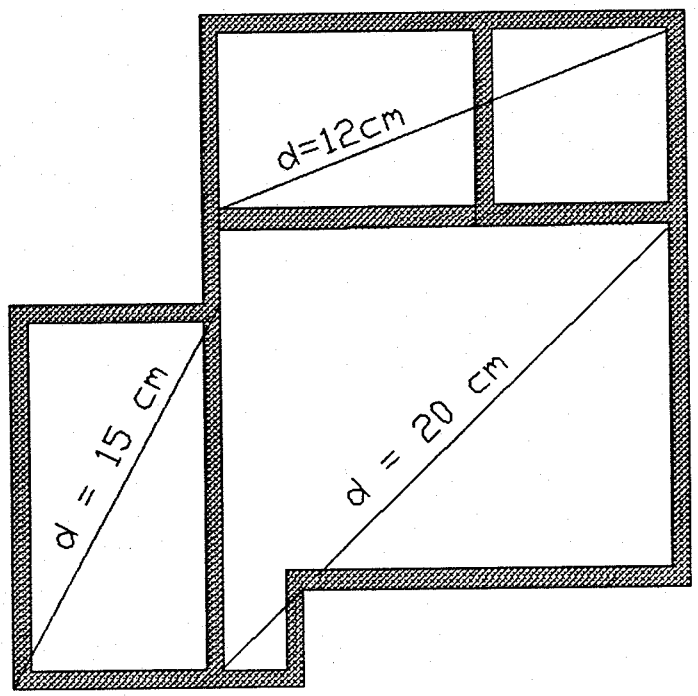


Fig. 3.4 A plate consisting of three different panels; panel = part of the plate with a uniform thickness

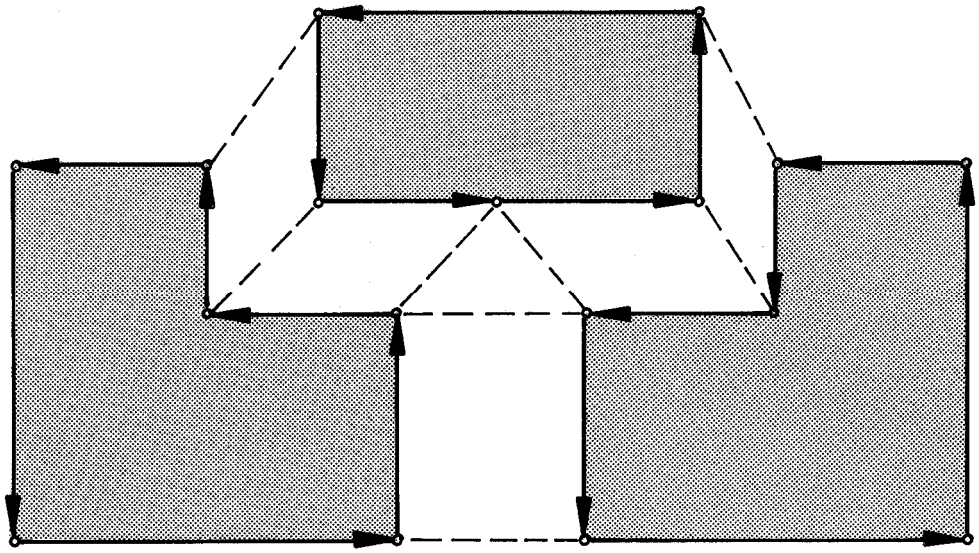


Fig. 3.5 In dialog input mode you encircle each panel with a closed polygon. At the interface of two panels the vertices of the two sides must match exactly, that is the end points must have the same coordinates.

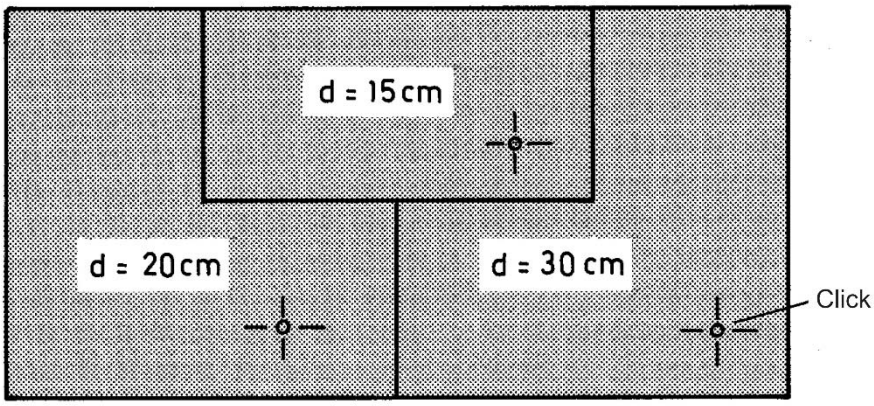


Fig. 3.6 In graphical input mode you can indicate the location of the single panels simply by clicking into the panels.

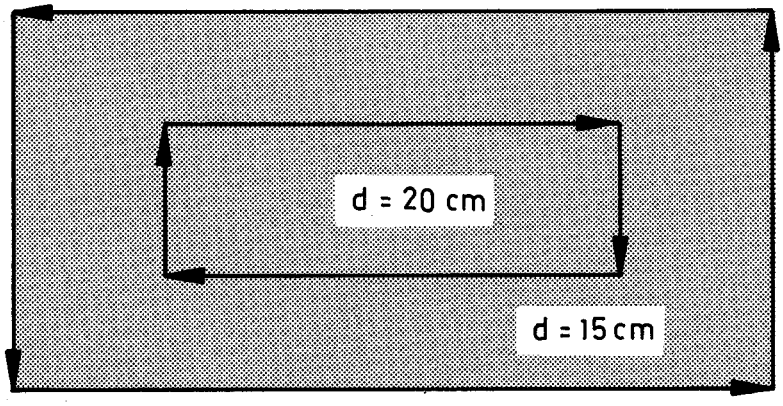
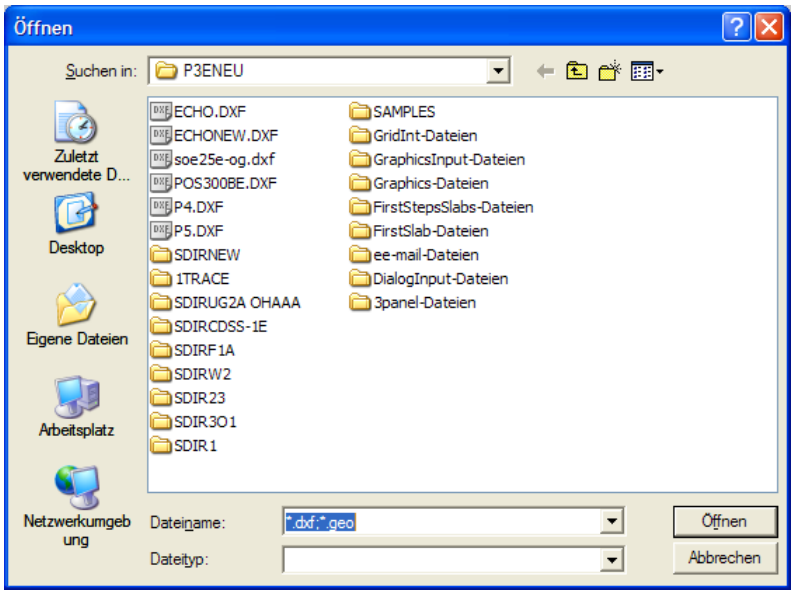


Fig. 3.7 Panels which are embedded into other panels are traversed in the opposite direction - opposite to the direction at the outer edge.

3.3. dxf-files

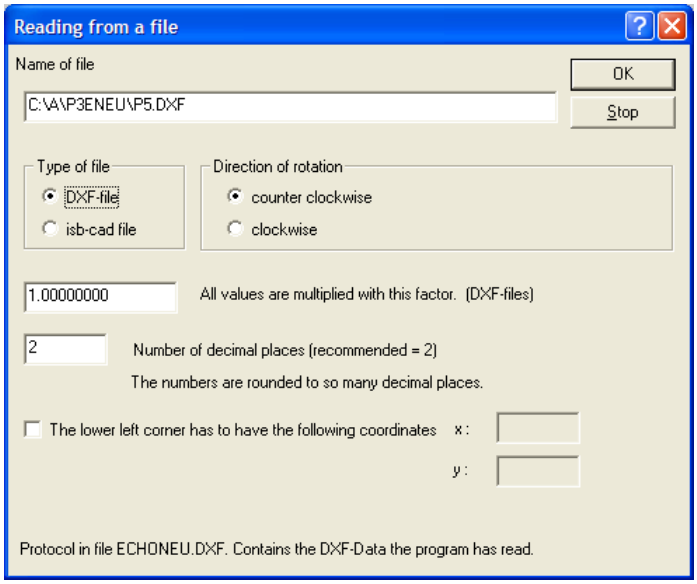
If the coordinates of the plate are stored in a dxf-file you can retrieve the coordinates from these files.

Click on the menu entry DXF and find the file.



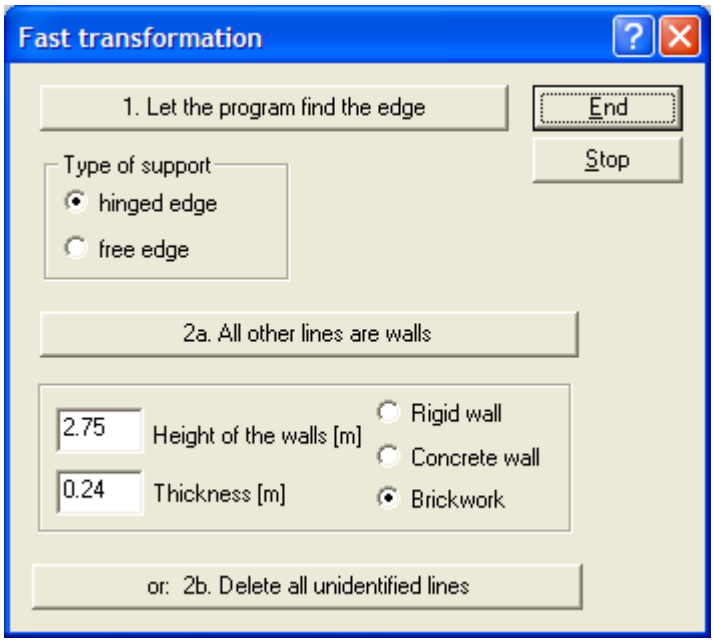
When the program retrieves the input from a DXF file it stores its interpretation of the input in an echo file. This allows you to check what the program has filtered out from the input stream.

The program will generate from the input stream the edge of the plate. To succeed it is important that the program knows the sense of rotation of the plate that is in which sequence the coordinates were stored in the dxf-file.



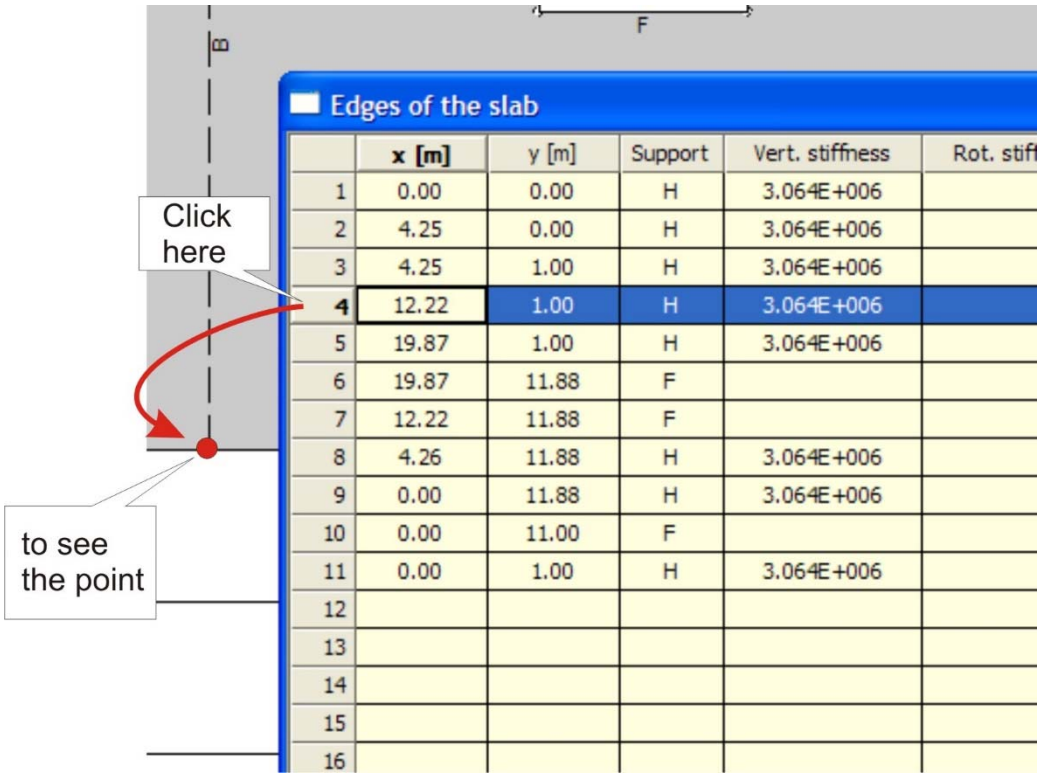
After the program has loaded the plate click on the icon with the magic wand

In the dialog, you can specify default actions to rapidly transform the line drawing into a plate.



First let the program find the edge of the plate. This is a test whether the outermost polygon ('the edge') is closed. Transform all other lines to walls; this can be changed later. Or delete all other lines to retain only the outer edge.

Additional details can be specified by working on the drawing as in graphical input mode.



	x [m]	y [m]	Support	Vert. stiffness	Rot. stif
1	0.00	0.00	H	3.064E+006	
2	4.25	0.00	H	3.064E+006	
3	4.25	1.00	H	3.064E+006	
4	12.22	1.00	H	3.064E+006	
5	19.87	1.00	H	3.064E+006	
6	19.87	11.88	F		
7	12.22	11.88	F		
8	4.26	11.88	H	3.064E+006	
9	0.00	11.88	H	3.064E+006	
10	0.00	11.00	F		
11	0.00	1.00	H	3.064E+006	
12					
13					
14					
15					
16					

3.4. Material

Units

The program works with the units Meter and Kilo Newton

Unit of length = m
Unit of forces = kN

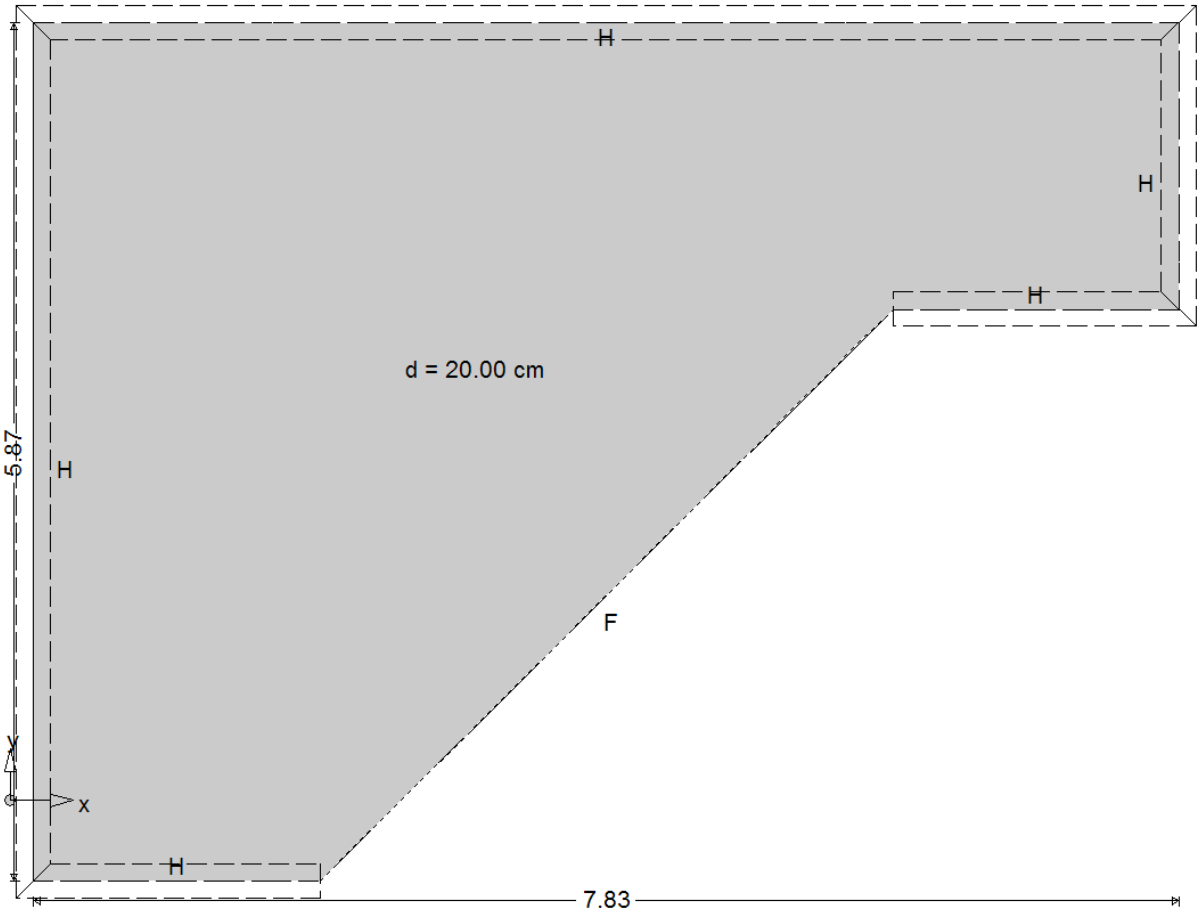


Fig. 3.8 The sides of an edge, H = hinged, F = free edge

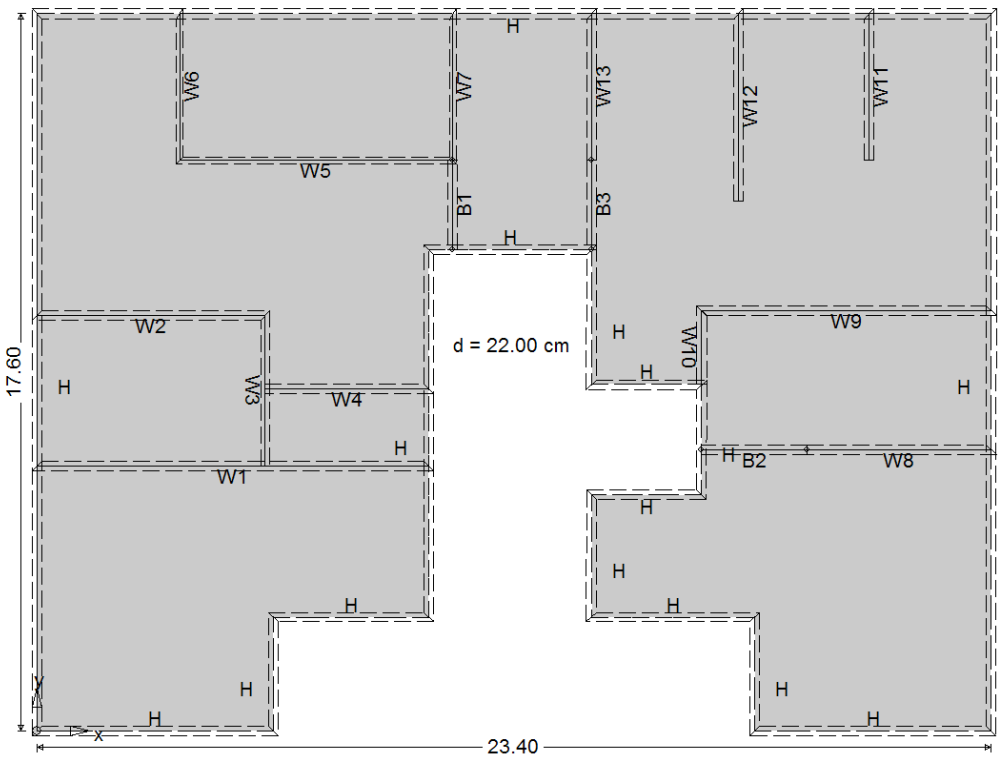


Fig. 3.9 A plate with hinged (H) sides and a series of interior walls (W) and three T-beams (B)

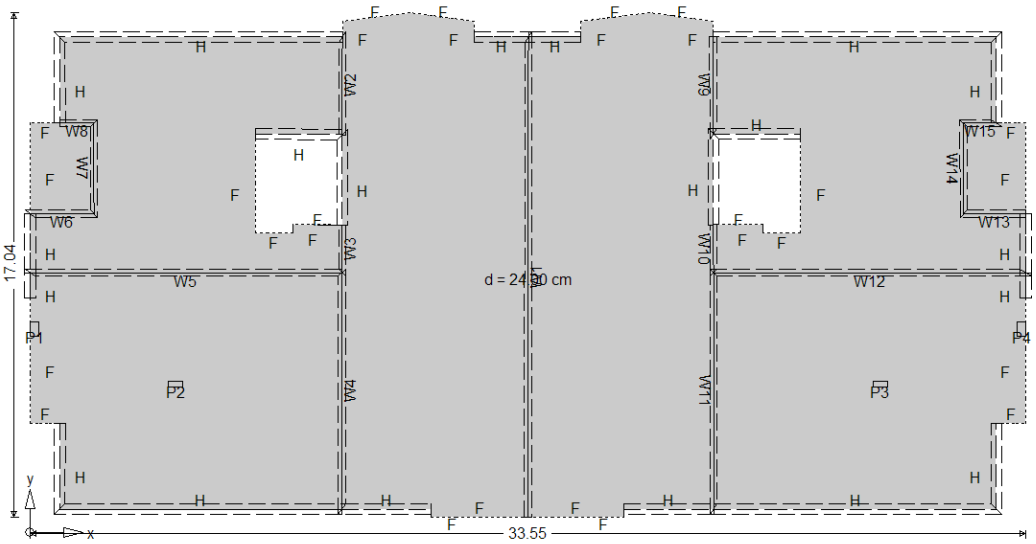


Fig. 3.10 Some sides of the two openings are unsupported free (F) and some sides are supported, H = hinged; P1, P2 and P3 are three piers, W = interior walls.

Modulus of elasticity

According to Euro-Code the modulus of elasticity of grade CX/Y concrete is as follows

C20/25	E = 2.49 * 10^7 kN/m^2
C25/30	E = 2.67
C30/37	E = 2.83
…	
C 0	E = defined by the user

To choose a grade C20/25 concrete enter C 20, for a grade C25/30 enter C 25, etc.

Slab thickness

	Concrete	E-Modulus [kN/m2]	Thickness [m]	Poisson's ratio
1	C 20	2.490E+007	0.220	0.16

A grade C 0 concrete signals the program that the user specifies the modulus of elasticity.

Poisson's constant

This constant assumes values in the range 0.1 --- 0.3. To compare a plate with beam solutions you should choose 0.0 for the constant.

Option: corners can lift

This option is only relevant at 90° corners where both sides of the plate are hinged. At other boundary conditions and other angles this option is ignored. If this option is activated the program allows the plate to lift on both sides of the boundary along a distance of 0.15 * l units, where l = length of the side.

Soft support

Soft support means that at skew angled hinged corner points the rotation – but not the deflection – is not hindered, that is the plate may rotate freely at such points.

This measure avoids the singularities in the support reactions that otherwise would develop at such points. As a rule, this switch should always be turned on.

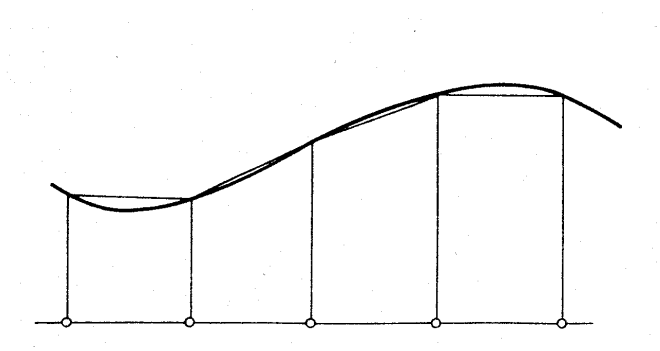


Fig. 3.11 Interpolation of a function with linear boundary elements

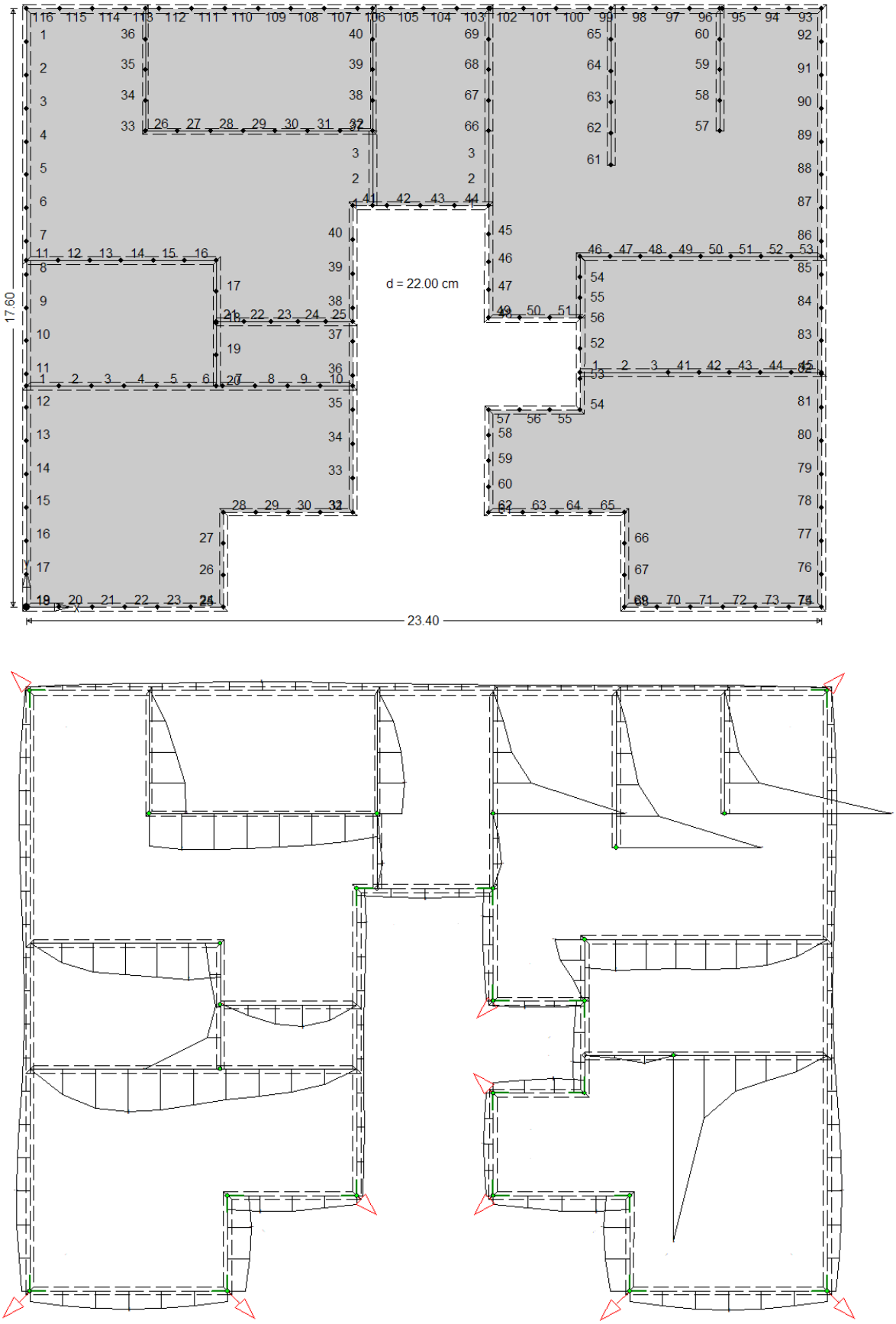


Fig. 3.12 The subdivision of the edges (and internal supports) into boundary elements allows a linear (or cubic – for w) interpolation of the support reactions and all the other unknowns along the edges.

The origin of the system of coordinates can lie at any point. It must not coincide with a corner point of the plate.

3.5. Panels

Each part of a plate with a uniform thickness is a panel. So, if the thickness of a plate is constant the plate consists of just one panel, s. Fig. 3.3, while a plate where the thickness in each of the three spans is different, as in Fig. 3.4, consists of three panels.

In dialog input mode, each panel is specified by entering the coordinates of the vertices of the edge of the panel. In graphic input mode, you mark the panel by clicking inside the panel; the program will then find the edge of the panel for you.

Graphical input mode

To start the input, click on **Specs (if graphical input)** and click on the bullet

Plate thickness

- changing

Next click on the button Mark panels where thickness is different and click on the single panels and input the thickness of these panels.

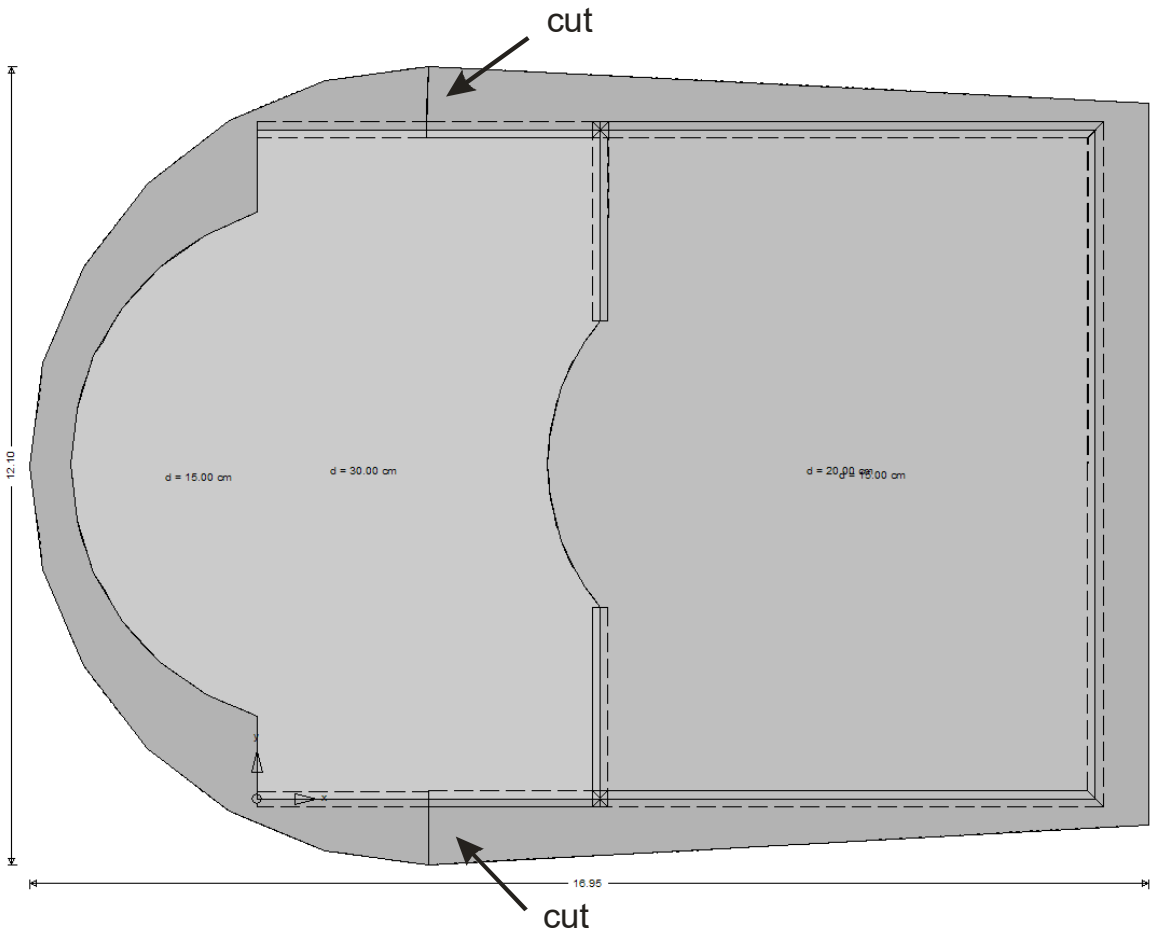


Fig. 3.13 A plate consisting of four panels. These panels form an array of panels, that is no panel is embedded into another panel. This is achieved by artificially subdividing the outer panel into two parts.

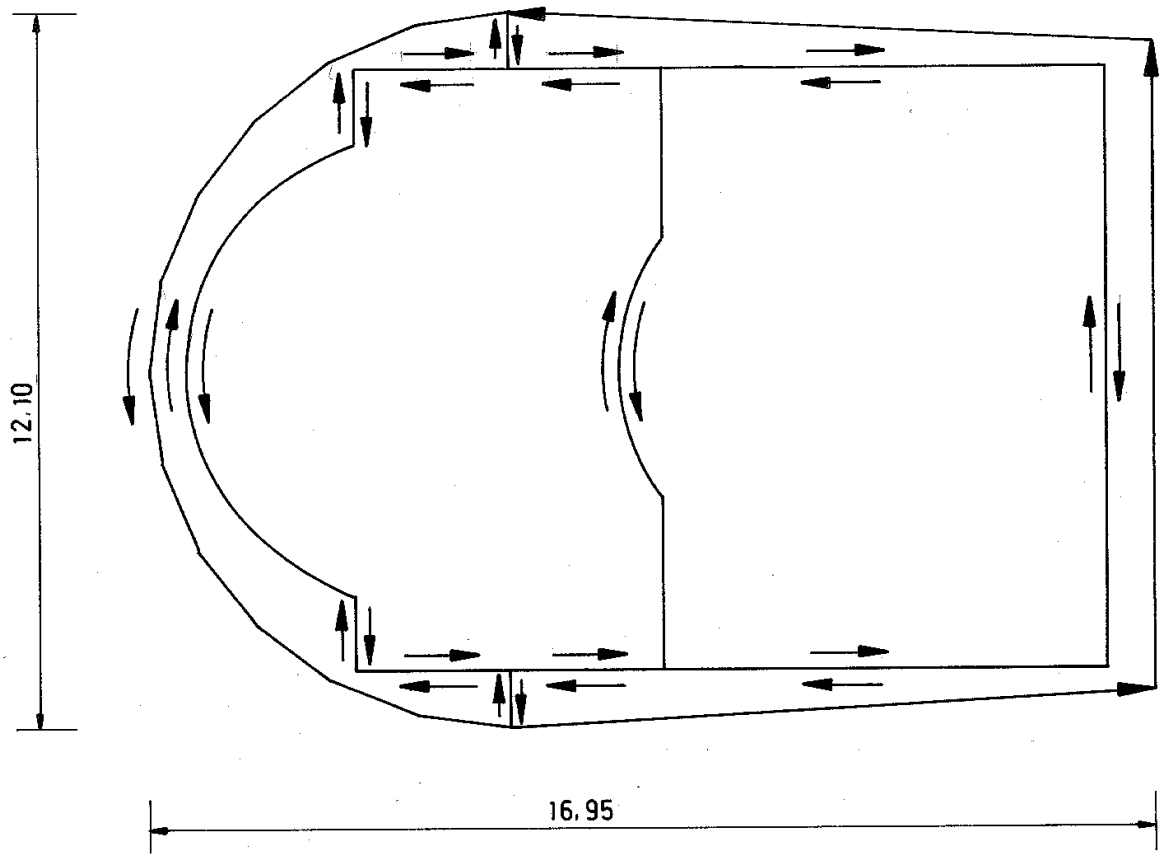


Fig. 3.14 The sense of rotation for each of the single panels is counter clockwise.

Dialog input

In dialog input mode, the shape of each panel is specified by entering the coordinates of **all** the vertices of the edge of the panel.

Make sure that the sides of neighboring panels coincide where the sides run in parallel. That is the coordinates of opposing end points must coincide. And make sure that the boundary conditions (or rather interface conditions) on both sides are the same. Either the boundary condition along the line that separates two panels is of type

interface	(= the thickness jumps) or
wall	(= the thickness jumps and a wall marks the border of the two panels)

Each such side appears twice in the input, because it belongs to the panel on the left side and to the panel on the right side.

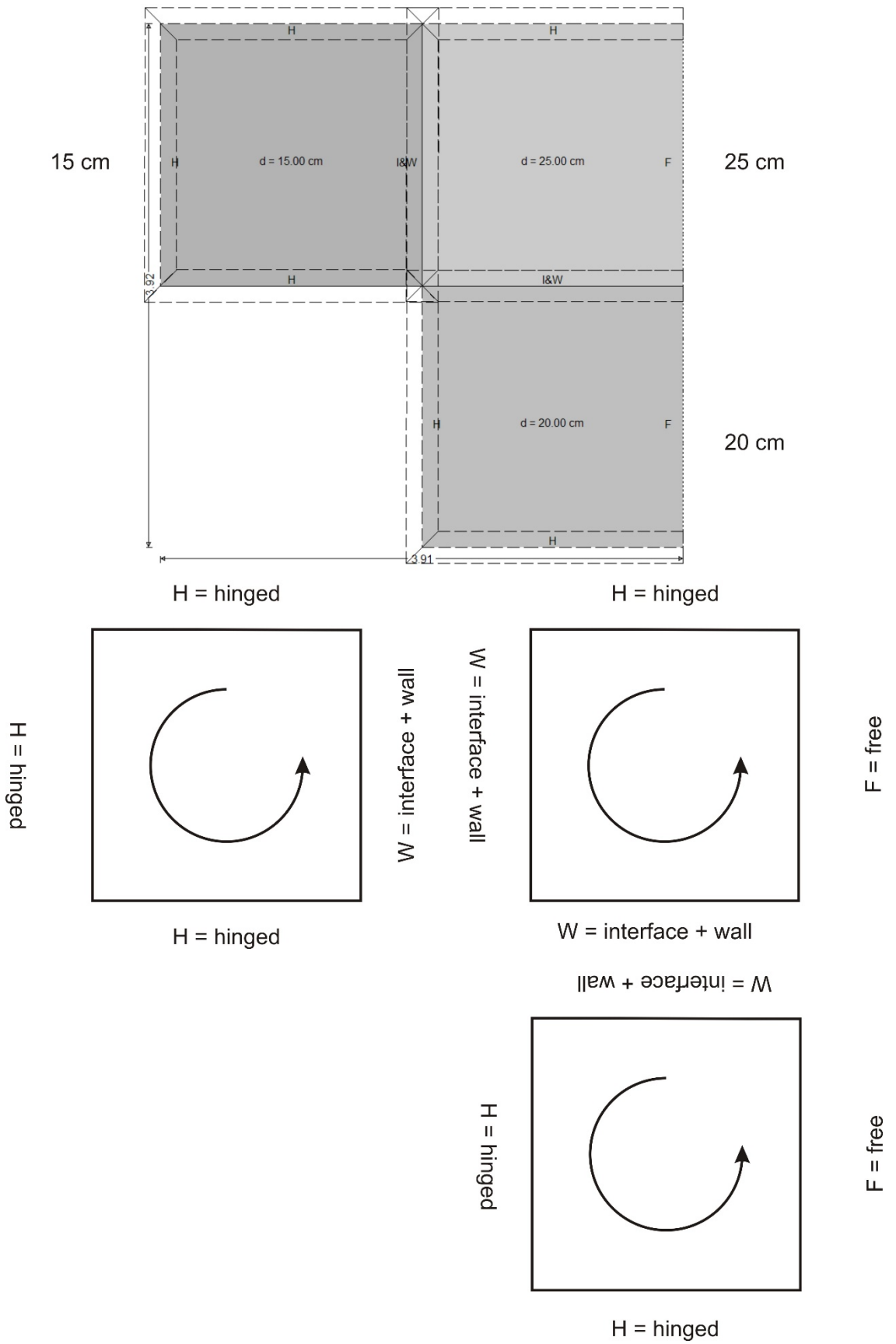


Fig. 3.15 A plate consisting of three panels.

Edges of the slab

	x [m]	y [m]	Support	Vert. stiffness	Rot. stiffness	Elements
1	0.16	3.36	H	2.618E+006		Autom.
2	2.12	3.36	W	2.618E+006		Autom.
3	2.12	5.32	H	2.618E+006		Autom.
4	0.16	5.32	H	2.618E+006		Autom.
5						

OK
Long input
Panel: 1

Edges of the slab

	x [m]	y [m]	Support	Vert. stiffness	Rot. stiffness	Elements
1	2.12	3.36	W	2.618E+006		Autom.
2	4.07	3.36	F			Autom.
3	4.07	5.32	H	2.618E+006		Autom.
4	2.12	5.32	W	2.618E+006		Autom.
5						

OK
Long input
Panel: 2

Edges of the slab

	x [m]	y [m]	Support	Vert. stiffness	Rot. stiffness	Elements
1	4.07	1.40	F			Autom.
2	4.07	3.36	W	2.618E+006		Autom.
3	2.12	3.36	H	2.618E+006		Autom.
4	2.12	1.40	H	2.618E+006		Autom.
5						

OK
Long input
Panel: 3

Fig. 3.16 In dialog input mode each of the above panels has four sides.

A problem are inclusions, that is if a panel is embedded into another panel. The sense of rotation along the edge of the inclusion is opposite to the sense of rotation of the outer edge of the plate. That is the edges of inclusions are traversed in the same sense as the edges of openings.
If the sense of rotation on the outer edge is counter clockwise then the sense of rotation for the panel is clockwise. The boundary condition on the edge of an inclusion is of type interface, because across the interface the thickness changes.

3.6. Edges

Each panel has an edge. If a plate consists of two panels then it has two edges. If in addition there is an opening, then the plate has three edges.

Number of edges = number of panels + number of openings

3.7. Sides

The sides of a plate form the edge of the plate, s. Fig. 3.8. Boundary conditions specify the support conditions of the single sides of the plate, see Fig. 3.9.

Possible boundary conditions are

clamped	Deflection and slope are zero
hinged	Deflection and bending moment are zero
free edge	Bending moment and support reactions are zero
symmetry axis	Slope and support reactions are zero
rotational spring	Bending moment and slope are coupled
translational spring	Deflection and support reactions are coupled
rotational and translational spring	

At the interface between two panels the boundary conditions can be of the following type

interface (= the thickness is discontinuous across the interface) or
wall (= as before plus the interface is supported by a wall)

So, the term *wall* taken as an interface condition means that along the line that separates the two panels a wall supports the plate (and that the thickness of the plate is different on both sides of the interface).

In graphical input mode, you can specify the boundary conditions of the single sides by clicking on the icon

by choosing the pertinent boundary conditions, and by then clicking on the single sides.

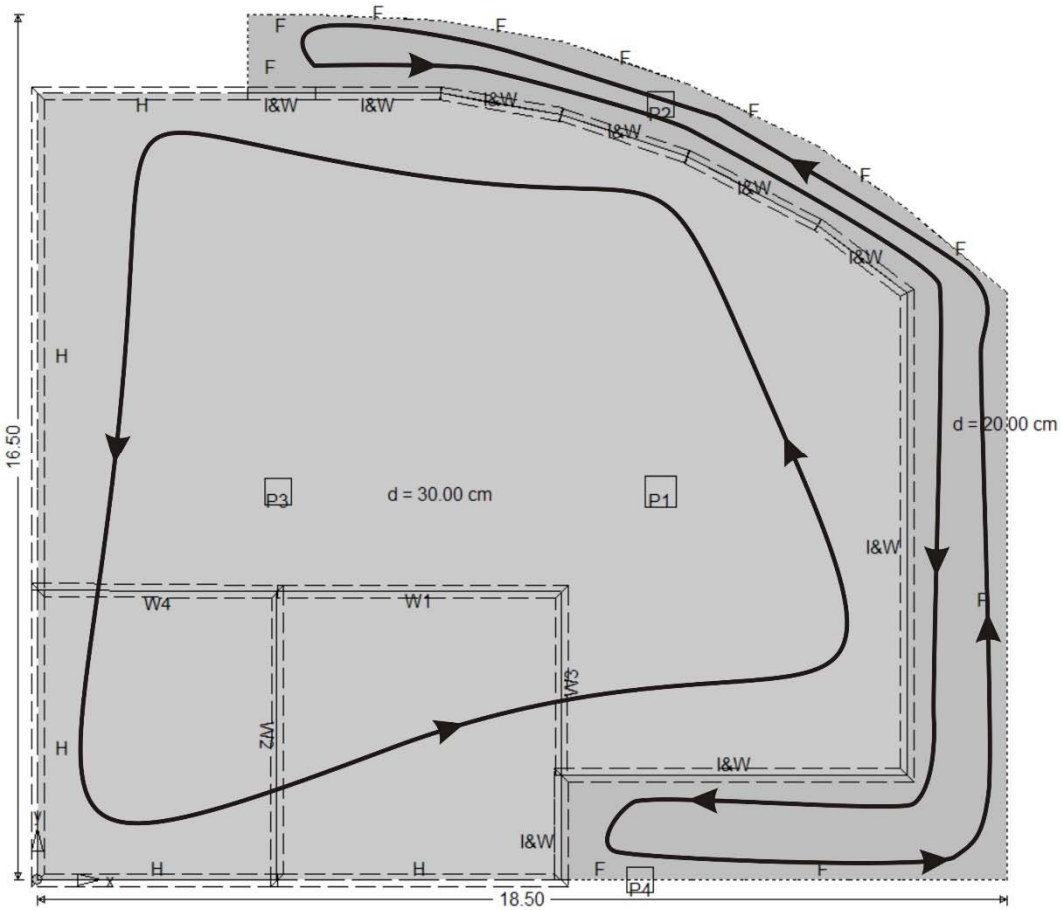


Fig. 3.17 Plate consisting of two panels; the sense of rotation along the edge of each of the two panels is counter clockwise

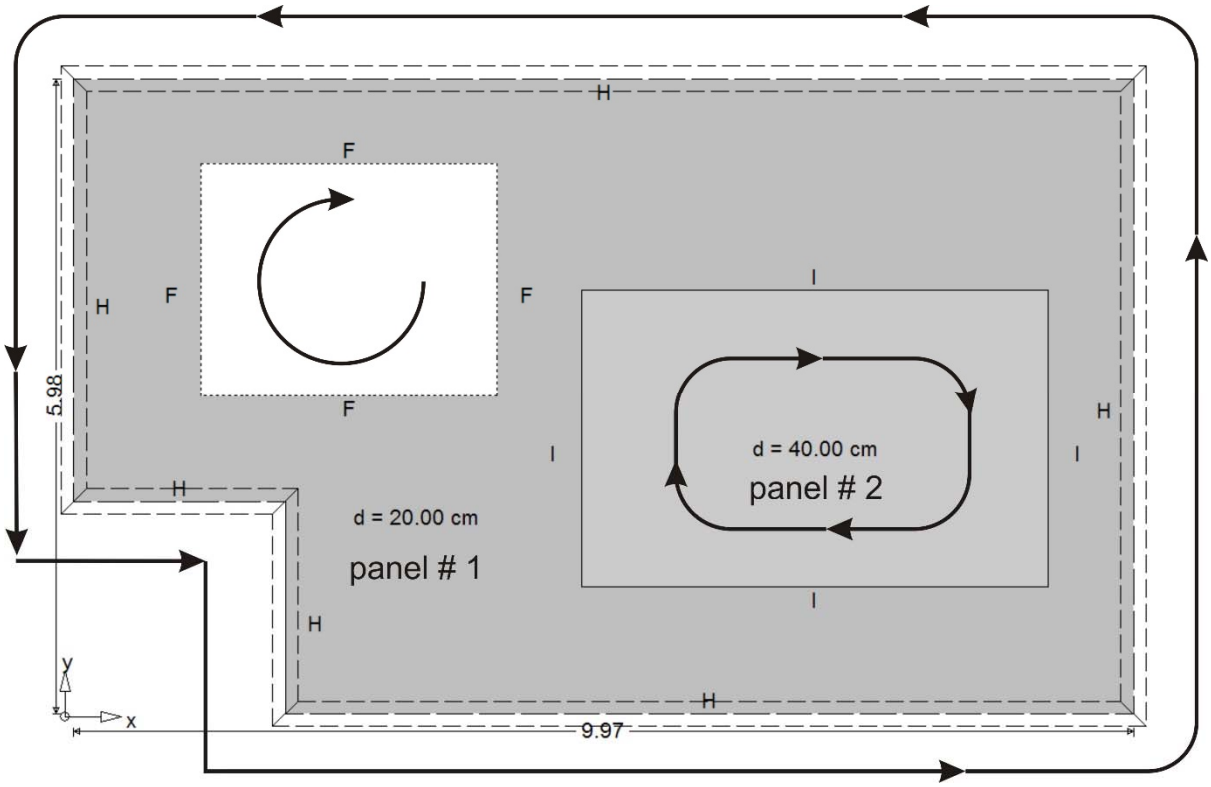


Fig. 3.18 The sense of rotation on the outer edge is counter clockwise if the y-axis points upward while it is clockwise along the edge of the opening and the edge of the panel # 2 which is embedded into panel # 1.

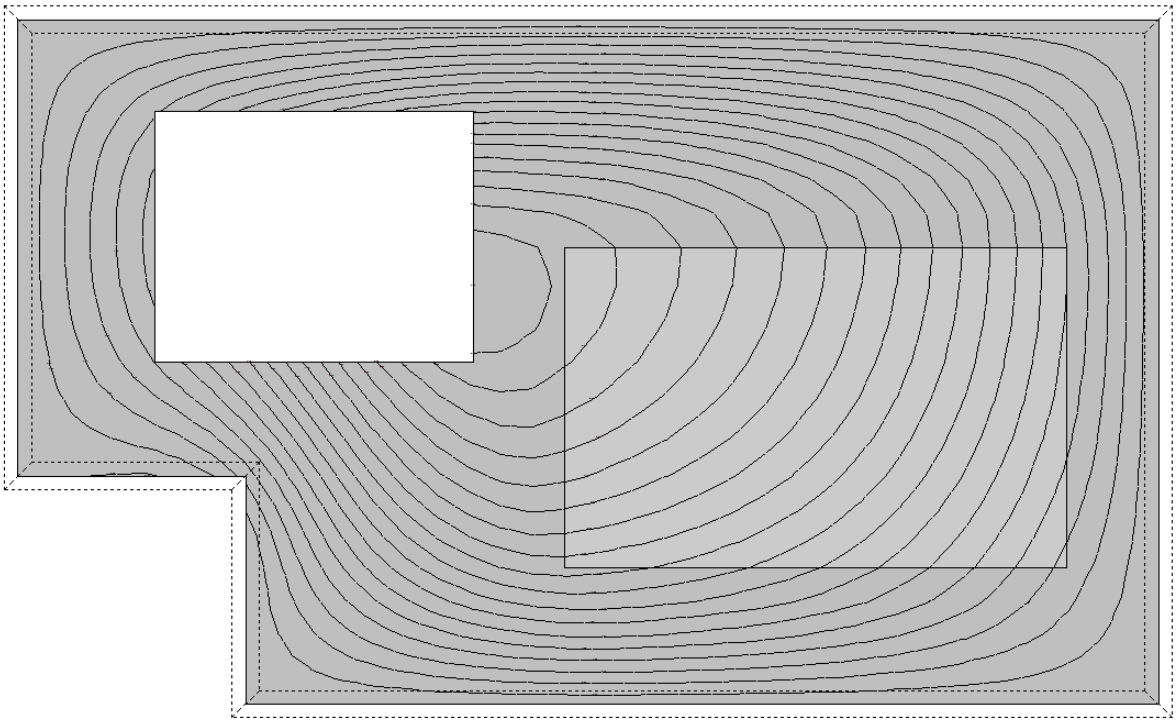
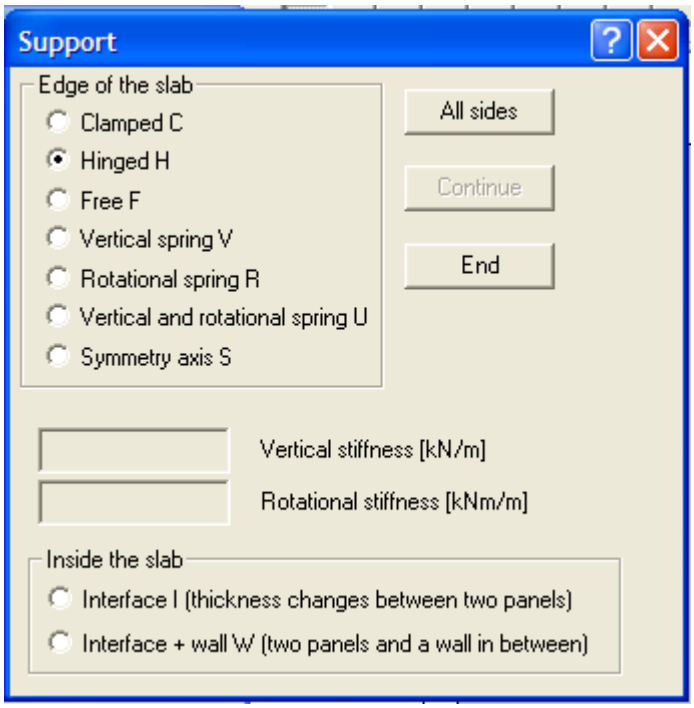


Fig. 3.19 Contour lines of the deflection under a uniform load.



The edges of foundations plates are usually unrestrained (free edge). But you can also restrain the edges with translational springs or rotational springs.

In the edge of the plate is supported by a T-beam the correct boundary condition for the edge is free because the edge can - to some degree - rotate freely and move up and down. So, the edge essentially is a free edge.

The T-beams are entered separately by clicking on the icon

The same is true in dialog input mode. The boundary condition of the edge is specified as free and, additionally, a T-beam which runs parallel to the edge of the plate is placed under the edge.

3.8. Boundary elements

The sides of the plate are subdivided into boundary elements, see Fig. 3.8, and the walls in the interior. T-beams are subdivided into finite elements. The subdivision is usually done by the program automatically but you can also do it 'manually', by specifying the number of elements for each side or wall yourself.

There is no simple rule for how many elements suffice. Theoretically the more elements you use the better results. For typical floor plates, the element length should be about 0.5 meter.

But it is not necessarily the absolute size of the elements which is important but rather the ratio of element length to the circumference of the plate. Very long and slender plates require a higher number of boundary elements than floor plates having a standard shape. (The optimal shape in this sense would be a quadratic plate).

Note that the boundary elements have no mechanical meaning, that they are just a mathematical tool to interpolate boundary functions along the edge of a plate.

3.9. Generating the boundary elements

The best approach is to choose an *average element length* and to let the program do the subdivision of the edge into boundary elements. Later you can manually change the number of elements along single sides. For this you must turn off the automatic subdivision.

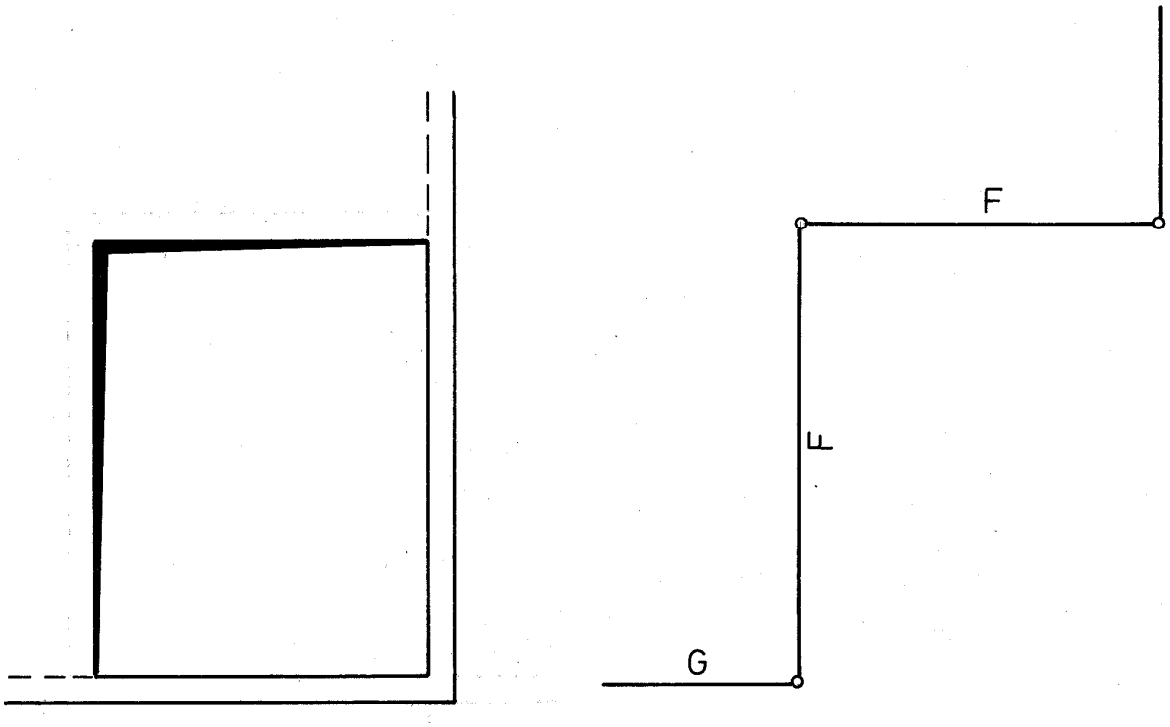


Fig. 3.20 Openings that border directly the edge of the plate are skipped. Instead the edge of the plate is moved inward.

3.10. Stiffness of supports and walls

The stiffness k (per unit length) of a support depends on the cross-section A of the wall and the modulus of elasticity E of the material and the height of the wall.

k = E A / height E = E-Modulus A = cross-sectional area = width * 1.0

The program knows three types of materials

Rigid material	E = 1.0 E 24	kN/m^2
Concrete	E = 3.0 E 7	kN/m^2
Brick work	E = 6.0 E 6	kN/m^2

with preset values for E, but you can overwrite these values and specify your own material. Eventually you must update the stiffness in the single dialogs (walls, edges) if you modify these preset values *after* you have input the plate.

3.11. Openings

Openings can have any shape and any kind of boundary condition is allowed for the sides of the opening. If a side rests on a wall then it has a hinged edge along that part of its edge, s. Fig. 3.14. Openings that border directly on the outer edge of the plate, see Fig. 3.15, should be avoided. Rather the edge of the plate should be drawn inward.

Graphical input mode

First, draw the edge of the opening, click on the icon

and mark the opening!

Dialog input mode

In dialog input mode, you enter the coordinates of the corner points of the opening. The sense of rotation is opposite to the sense of rotation on the outer edge of the plate.

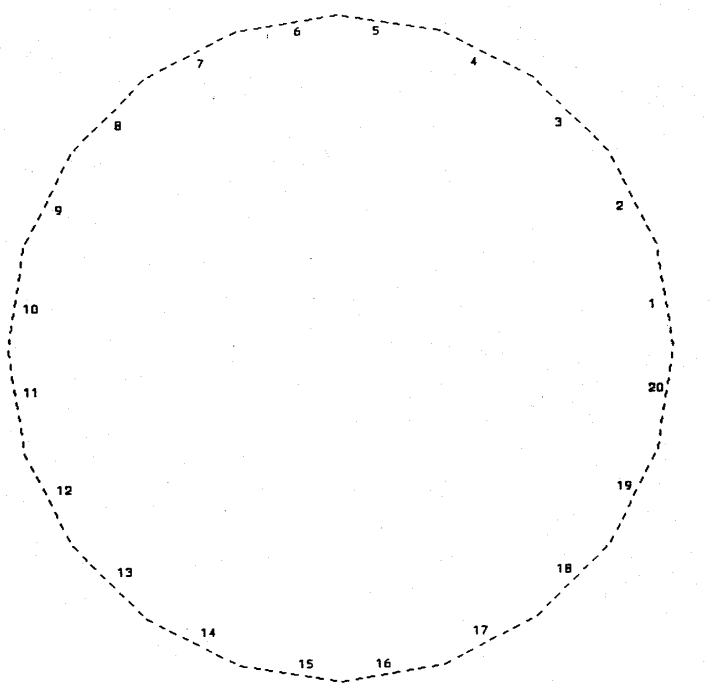


Fig. 3.21 Polygonal approximation of the edge of a circular plate.

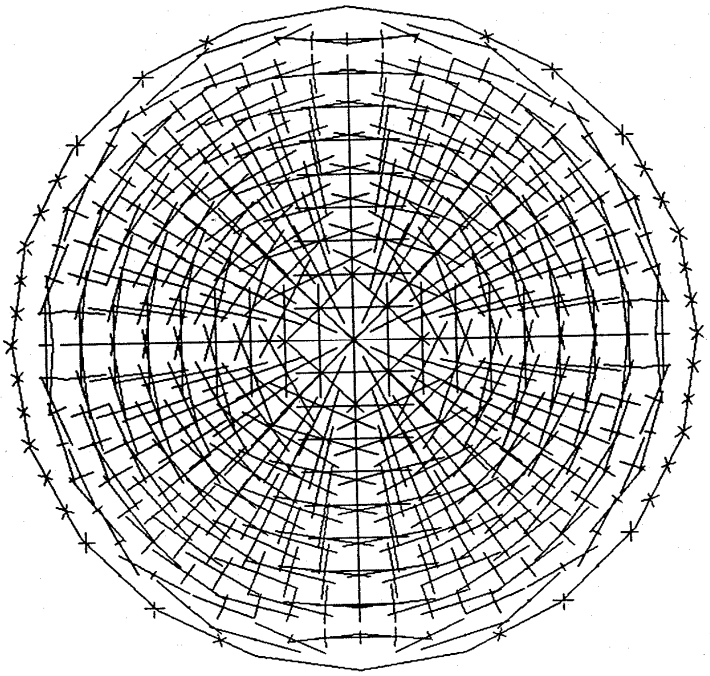


Fig. 3.22 Principal moments under uniform load

3.12. Circular plates

The edge of a circular plate is approximated by a polygon as in Fig. 3.12 and 3.13. If the edge of the plate is hinged you should allow at least provide three elements on each straight side of the polygon.

Graphical input mode

Click on the icon

3.13. Circular plates with a circular opening (annulus)

Such plates can be generated easily by clicking on the icon

3.14. Circular arcs

In graphical input mode, you can specify such sides in two ways:

You specify two points and the center point of the circular arc

or you specify three points on the arc

In dialog input mode, you can enter a curved edge as follows:
Click on Edge and choose Long Input. The shape of the edge can be

S = straight
C = curved

If the side is curved a positive radius means that the side is convex and a negative radius that the side is concave.

The arc extends from the starting point of the side up to the beginning of the next side.

3.15. Interior walls

Walls standing directly under the edge of the plate and parallel to the edge of the plate (edge supports) are not entered separately. They provide the support conditions for the plate

hinged, clamped, vertical spring, rotational spring, etc.

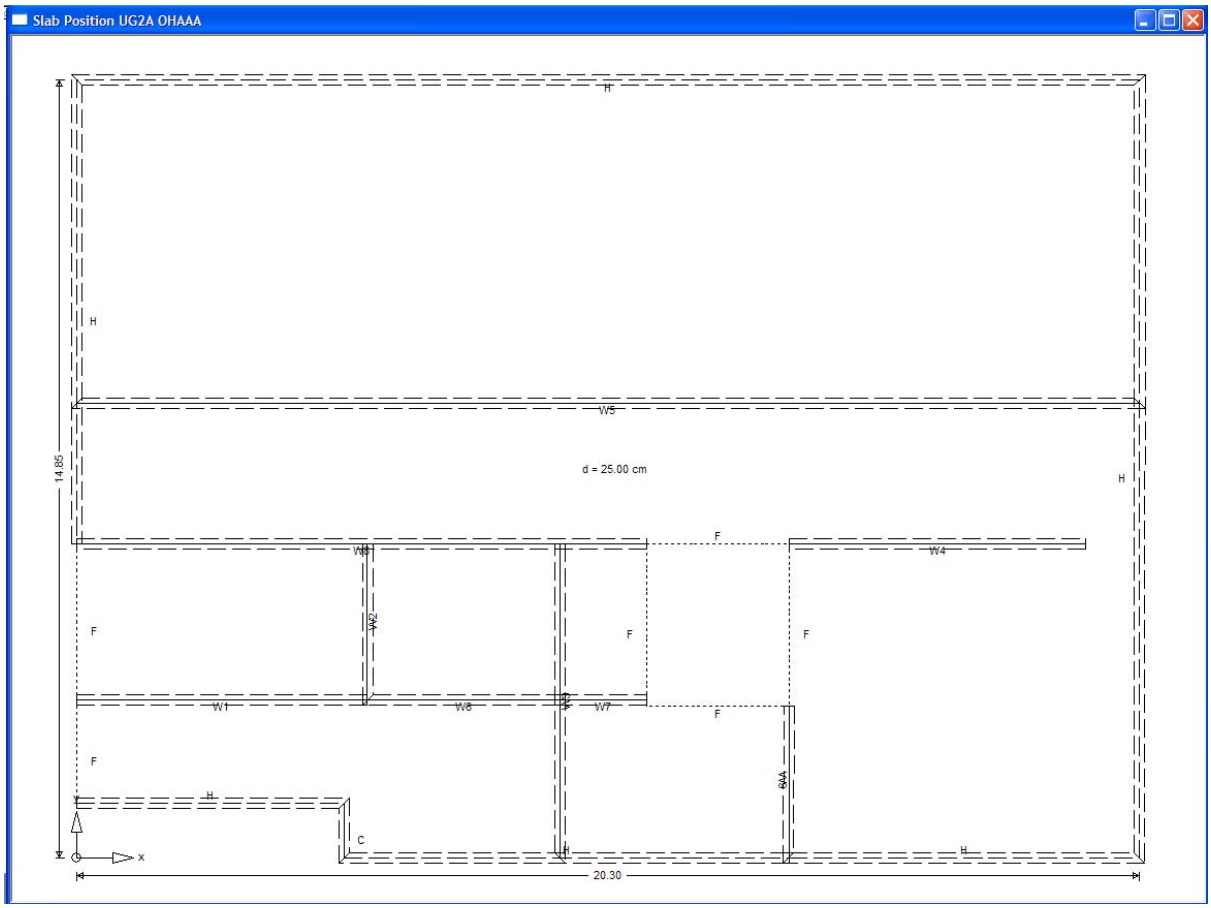
and therefore, it is not necessary to input these walls. Only the walls in the interior are entered as such.

Walls in the interior which separate two panels of the plate (panel = various thickness) are - according to the logic of the program – not interior walls but part of the edge of the single panels.
Hence, they are input where the edge of the single panels is detailed in the input. That is each panel is encircled by an edge and the single sides of these edges are specified by entering the coordinates of the vertices and by entering the support conditions of the single sides. The support condition (better interface condition) for such walls between two panels is 'wall'.

In graphical input mode, interior walls are entered by first clicking on the icon

and next on all the single walls.

When you click on the first wall a context menu will open where you can specify the building material, the modulus of elasticity etc. This menu stays close when you click on the next walls. To activate this menu again to change the preselect ion press the right mouse button.



Interior walls

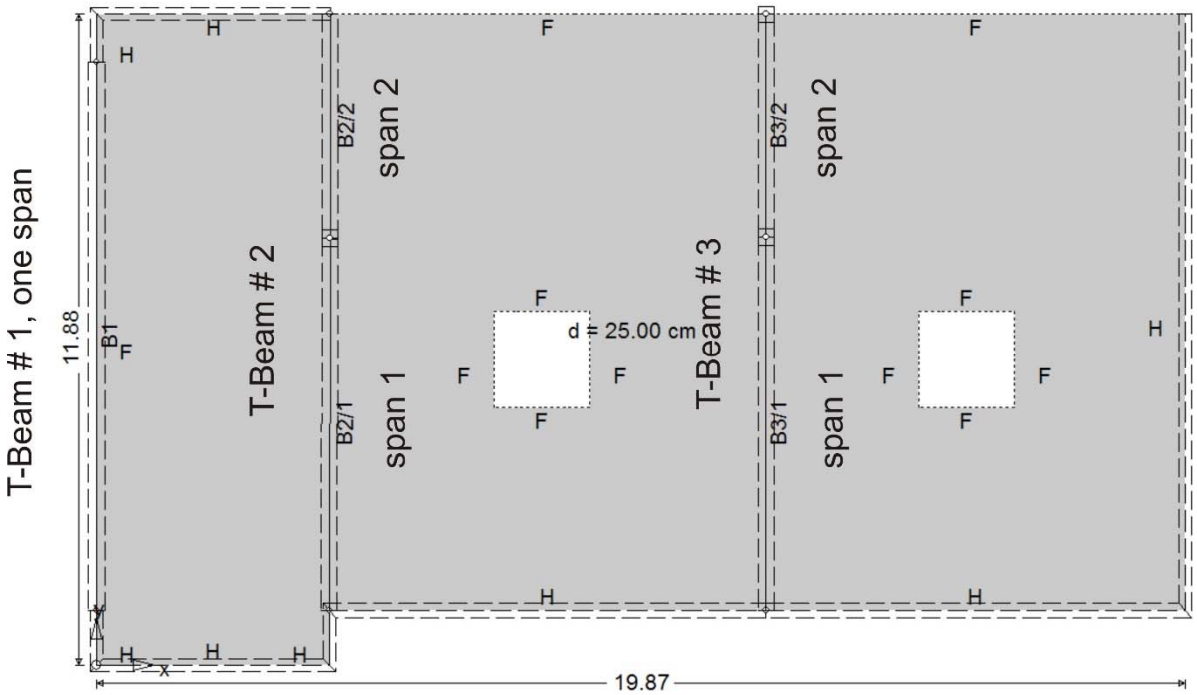
Nr.	from x [m]	from y [m]	to x [m]	to y [m]	Length [m]	Height [m]	Thick. [m]	E-Modulus [kN/m2]	Material
1	0.00	3.02	5.56	3.02	5.56	2.97	0.20	6.00E+006	Bricks
2	5.56	3.02	5.56	6.00	2.98	2.97	0.20	6.00E+006	Bricks
3	9.23	0.00	9.23	6.00	6.00	2.97	0.20	6.00E+006	Bricks
4	13.62	6.00	19.27	6.00	5.65	2.97	0.20	6.00E+006	Bricks
5	0.00	8.68	20.30	8.68	20.30	2.97	0.20	6.00E+006	Bricks
6	5.56	3.02	9.23	3.02	3.67	2.97	0.20	6.00E+006	Bricks
7	9.23	3.02	10.90	3.02	1.67	2.97	0.20	6.00E+006	Bricks
8	10.90	6.00	0.00	6.00	10.90	2.97	0.20	6.00E+006	Bricks
9	13.60	2.90	13.60	0.00	2.90	2.97	0.20	6.00E+006	Bricks

3.16. Beams

Beams are modeled by a series of beam elements. The stiffness of the beams depends on the effective width of the beam and the dimensions of the web. The number of spans of a beam is limited to 10.

Graphical input of beams

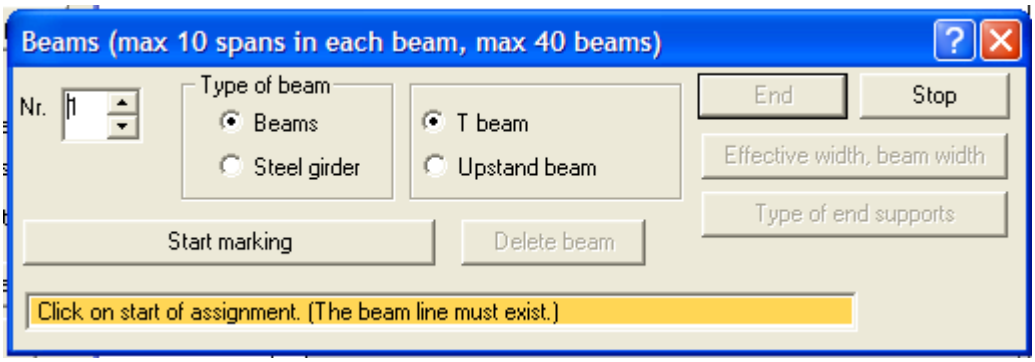
Split the beam into single but continuous lines. One line for each span. If possible, the beam should be straight and should not change its direction.




```
Beams

Beam Span Web thickness [m] height [m]       Effective width  [m]   EI [kNm2]
-------------------------------------------------------------------------------
 1   1       0.30          0.30                 0.60              1.713E+005

 2   1       0.30          0.30                 0.60              1.713E+005
 2   2       0.30          0.30                 0.75              1.875E+005

 3   1       0.30          0.30                 0.60              1.713E+005
 3   2       0.30          0.30                 0.75              1.875E+005

Spans of the beams
------------------

Upstand beam  1  vertical     1 span                                End support

supp.  1  x =   0.00 y =   1.00    on  Wall                       hinged
supp.  2  x =   0.00 y =  11.00    on  Wall                       hinged

Upstand beam  2  vertical     2 spans

supp.  1  x =   4.25 y =   1.00    on  Wall                       hinged
supp.  2  x =   4.26 y =   7.78    on  Pier    0.30/0.30/2.75
supp.  3  x =   4.26 y =  11.88    on  Wall                       hinged

Upstand beam  3  vertical     2 spans

supp.  1  x =  12.22 y =   1.00    on  Wall                       hinged
supp.  2  x =  12.22 y =   7.80    on  Pier    0.30/0.30/2.75
supp.  3  x =  12.22 y =  11.88    on  Pier    0.30/0.30/2.75    hinged
```

After you have drawn the single lines representing the single spans, click on the icon

and next mark the end points of the single spans by beginning with the left-most support or end of the beam. Click at each intermediate support in between (the end points of the single spans) and finally on the right-most support or end of the beam.



Is

Next you must specify the effective width and the dimensions of the web of the beam. And the type of end support of the beam

F = Free *H = hinged* *C = clamped* *E = rotational spring*

The intermediate support of a beam (the support of the end points of the spans) can be an interior wall, a pier or another beam which intersects the beam. If the intermediate support is a wall or a pier the situation is simple.

But if the intermediate support comes from a second beam, that is if two beams intersect at this point, the situation can be the following:

i) the two beams provide mutual support to each other (no external support)
ii) a pier is placed under the intersection so that the two beams rest on the pier

In the first case, the support condition at the intersection is for *both beams* of type 'B' = beam.

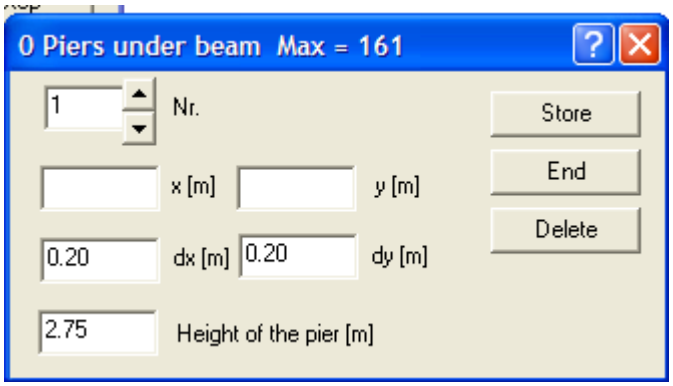
In the second case, the support condition for **one** beam is 'B' = beam and **for the other** beam it is 'P' = pier. The 'P' is reserved for the beam with the pier as the intermediate support and the 'B' for the beam that intersects this beam. The one with the 'P' is the beam where you specify a pier as intermediate support by providing values of the cross section and the length of the pier.

The program distinguishes between standard piers and piers under beams. The latter form only an *indirect* support of the plate. They provide end or intermediate supports for a beam. As such they are only entered when it comes to specifying the intermediate supports.

Please do not place a standard pier under a beam.

In graphical input mode click on the icon

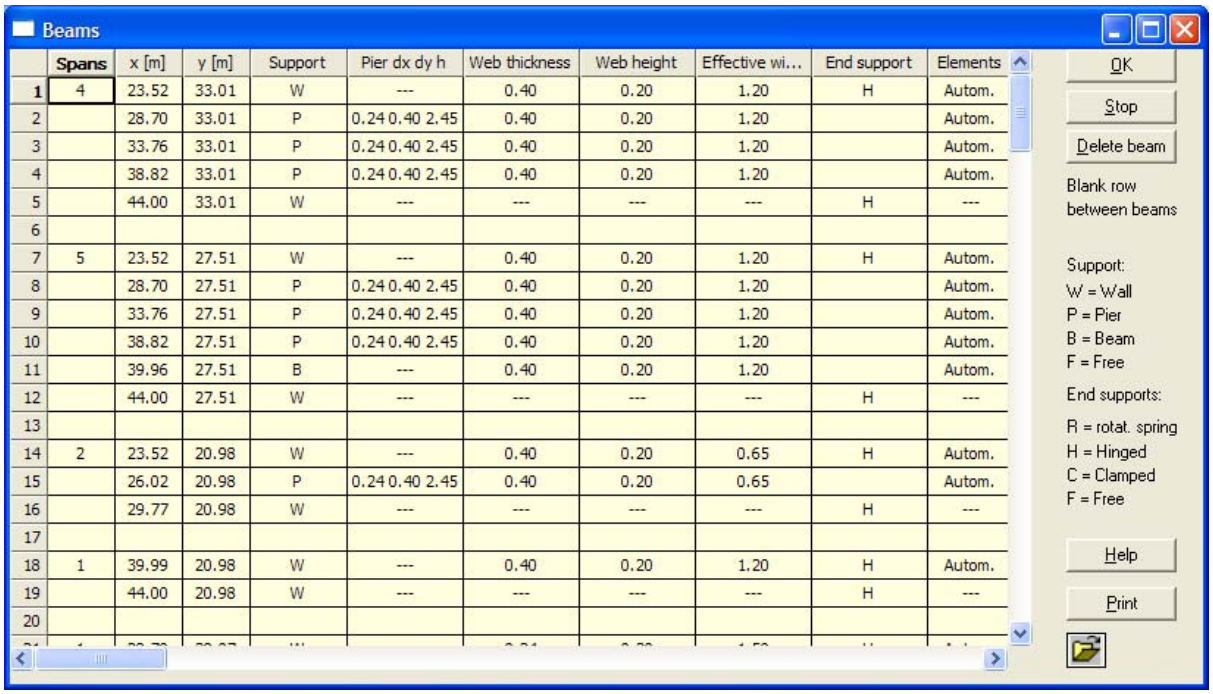
and mark the position of these intermediate pier supports with a mouse click. In the context menu that opens you can specify the shape of the pier.



On the dimensions of the pier depends the stiffness of the intermediate support.

Dialog input of T-beams

T-beams are input as in the following table:



Beams

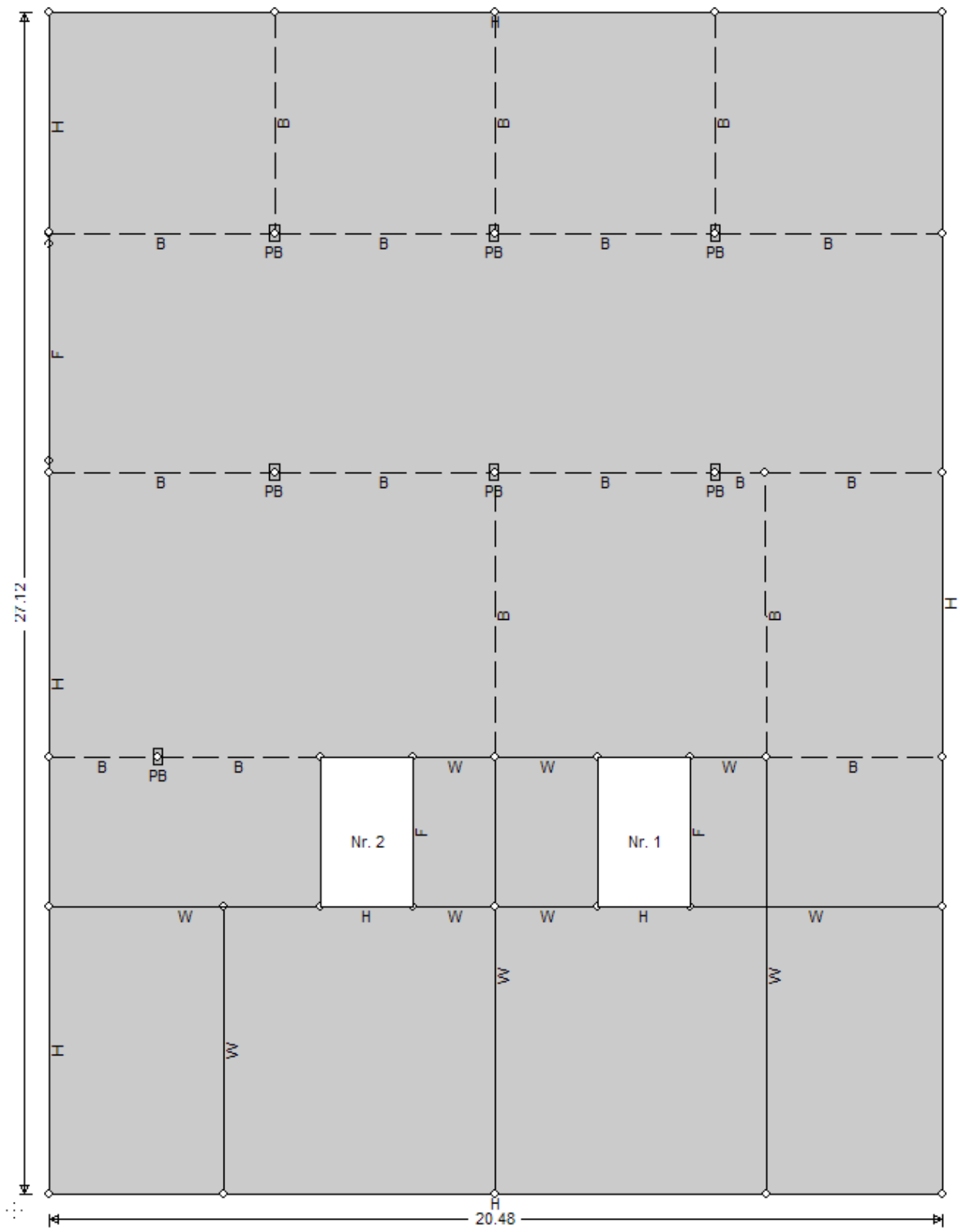
	Spans	x [m]	y [m]	Support	Pier dx dy h	Web thickness	Web height	Effective wi...	End support	Elements
1	4	23.52	33.01	W	---	0.40	0.20	1.20	H	Autom.
2		28.70	33.01	P	0.24 0.40 2.45	0.40	0.20	1.20		Autom.
3		33.76	33.01	P	0.24 0.40 2.45	0.40	0.20	1.20		Autom.
4		38.82	33.01	P	0.24 0.40 2.45	0.40	0.20	1.20		Autom.
5		44.00	33.01	W	---	---	---	---	H	---
6										
7	5	23.52	27.51	W	---	0.40	0.20	1.20	H	Autom.
8		28.70	27.51	P	0.24 0.40 2.45	0.40	0.20	1.20		Autom.
9		33.76	27.51	P	0.24 0.40 2.45	0.40	0.20	1.20		Autom.
10		38.82	27.51	P	0.24 0.40 2.45	0.40	0.20	1.20		Autom.
11		39.96	27.51	B	---	0.40	0.20	1.20		Autom.
12		44.00	27.51	W	---	---	---	---	H	---
13										
14	2	23.52	20.98	W	---	0.40	0.20	0.65	H	Autom.
15		26.02	20.98	P	0.24 0.40 2.45	0.40	0.20	0.65		Autom.
16		29.77	20.98	W	---	---	---	---	H	---
17										
18	1	39.99	20.98	W	---	0.40	0.20	1.20	H	Autom.
19		44.00	20.98	W	---	---	---	---	H	---
20										

In the first column, you specify the number of spans. Next follow the x- and y-coordinates of the left end of the beam and the kind of support at the left end. The support can be either of type

F = free W = wall U = beam P = pier

where F = free means no support (cantilever), W = means that the end is supported by a wall, U = means that the support is provided by a second beam (the two beams intersect at this point) and P = means that the end of the beam is supported by a pier.

In the next column, you specify the dimensions of the pier. Next enter the web thickness, web height (the part of the beam below the plate) and the effective width of the T-beam. These values serve to calculate effective beam stiffness EI. Finally, you specify the support conditions for the left end of the beam. The ends can be restrained by a rotational spring (R), can be simply supported (H), can be clamped (C) or can be free to move (F). The same information must be provided also in the last row where the type of support at the right end of the beam is specified.


27.12
20.48
Nr. 1
Nr. 2
PB
B
H
F
W
VW

```
Beams

 Beam Span Web thickness [m] height [m]       Effective width  [m]   EI [kNm2]
 ---------------------------------------------------------------------------
  1   1       0.40          0.20                 1.20              1.040E+005
  1   2       0.40          0.20                 1.20              1.040E+005
  1   3       0.40          0.20                 1.20              1.040E+005
  1   4       0.40          0.20                 1.20              1.040E+005

  2   1       0.40          0.20                 1.20              1.040E+005
  2   2       0.40          0.20                 1.20              1.040E+005
  2   3       0.40          0.20                 1.20              1.040E+005
  2   4       0.40          0.20                 1.20              1.040E+005
  2   5       0.40          0.20                 1.20              1.040E+005

  3   1       0.40          0.20                 0.65              8.043E+004
  3   2       0.40          0.20                 0.65              8.043E+004

  4   1       0.40          0.20                 1.20              1.040E+005

  5   1       0.24          0.30                 1.50              1.551E+005

  6   1       0.24          0.30                 1.50              1.551E+005

  7   1       0.24          0.30                 1.50              1.551E+005

  8   1       0.24          0.40                 0.75              2.114E+005

  9   1       0.40          0.20                 2.00              1.280E+005

 10   1       0.40          0.20                 2.00              1.280E+005

 Spans of the beams
 ------------------

 Beam    1  horizontal   4 spans                            End support

 supp.  1  x =  23.52 y =  33.01    on  Wall                      hinged
 supp.  2  x =  28.70 y =  33.01    on  Pier    0.24/0.40/2.45
 supp.  3  x =  33.76 y =  33.01    on  Pier    0.24/0.40/2.45
 supp.  4  x =  38.82 y =  33.01    on  Pier    0.24/0.40/2.45
 supp.  5  x =  44.00 y =  33.01    on  Wall                      hinged

 Beam    2  horizontal   5 spans

 supp.  1  x =  23.52 y =  27.51    on  Wall                      hinged
 supp.  2  x =  28.70 y =  27.51    on  Pier    0.24/0.40/2.45
 supp.  3  x =  33.76 y =  27.51    on  Pier    0.24/0.40/2.45
 supp.  4  x =  38.82 y =  27.51    on  Pier    0.24/0.40/2.45
 supp.  5  x =  39.96 y =  27.51    on  T beam
 supp.  6  x =  44.00 y =  27.51    on  Wall                      hinged
```

3.16.1. Steel beams

The input of steel beams follows essentially the same pattern as that of T-beams.

```
Steel girders

 Beam span   E-Modulus [kN/m^2] moment of inertia I [cm^4]  EI [kNm2]
 ---------------------------------------------------------------
  1   1        2.100E+008        1.234E+004              2.591E+004

 Spans of the beams
 ------------------

 Beam    1  horizontal   1 span                              End support

 supp.  1  x =   0.00 y =   2.00    on  Wall                      hinged
 supp.  2  x =   5.00 y =   2.00    on  Wall                      hinged
```

The modulus of elasticity is set to E = 2.1 * 10^8 kN/m**²**. The user specifies the value of I, the moment of inertia of the cross section of the steel beam. Eventually different values for E can be considered by choosing

appropriate values for I because the effective parameter, EI, is a product of E and I. Because it is only EI which counts there is no difference internally between a T-beam and a steel beam.

3.16.2. Support reactions in the beams

T-beams and steel beams are modeled by a series of beam elements which are connected with the plate at the nodes. The (linear) support reactions [kN/m] of the beam elements are subject to the condition that the deflection of the beam elements and the plate are the same at the nodes. The beams have no torsional rigidity.

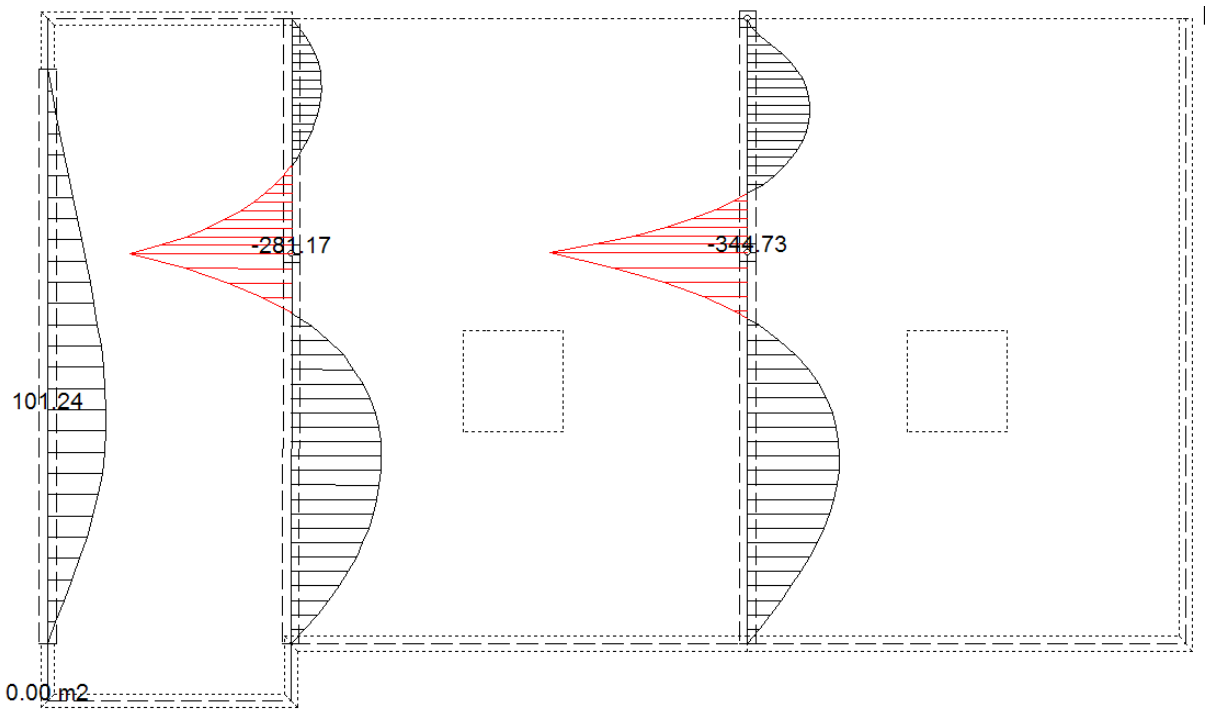


Fig. 3.23 Bending moments in beams

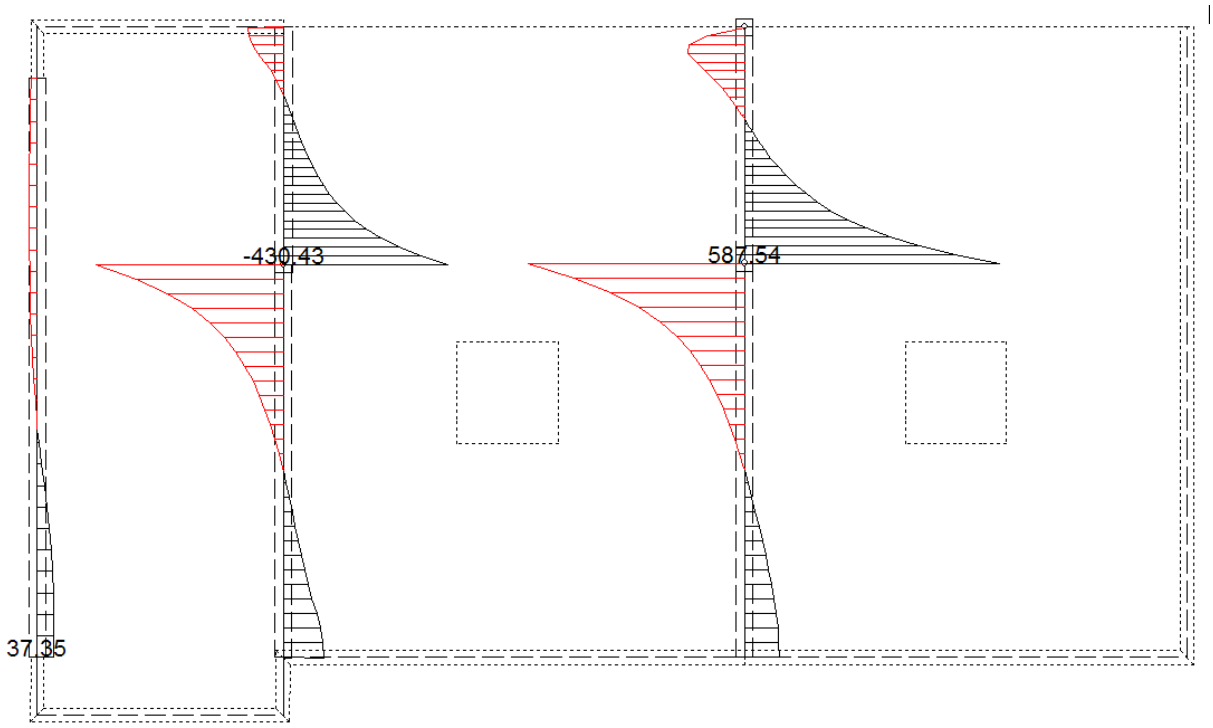


Fig. 3.24 Shear forces in the beams
These support reactions act as load on the beams and determine the internal actions in the beam.

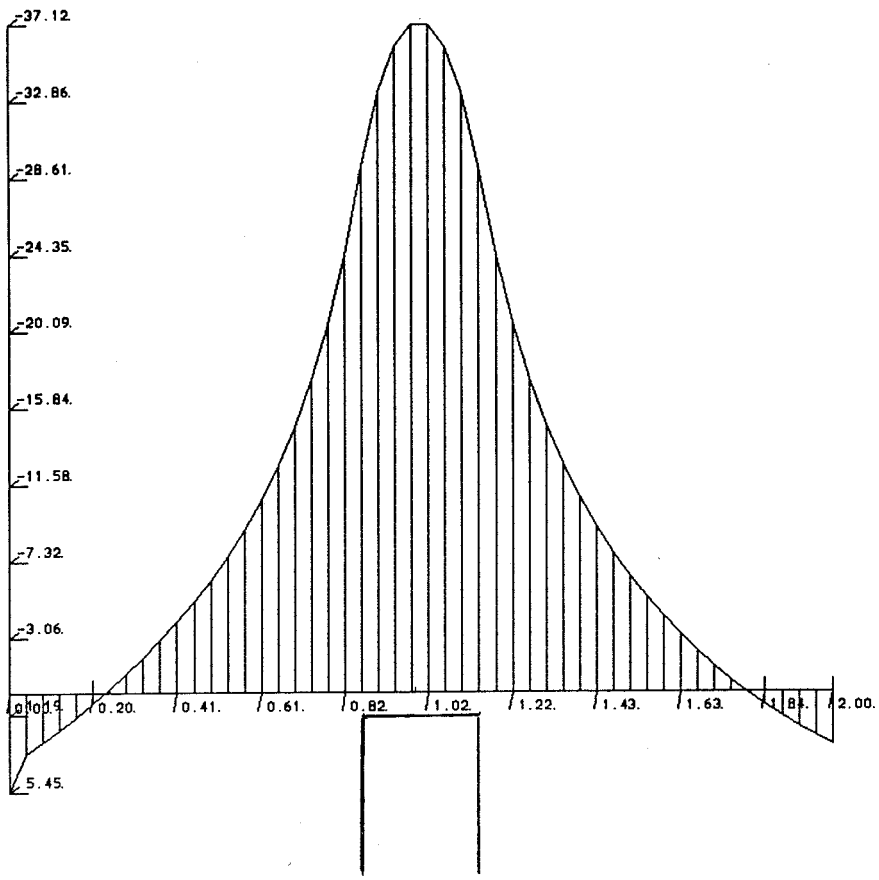


Fig. 3.25 Bending moments are automatically rounded out across piers.

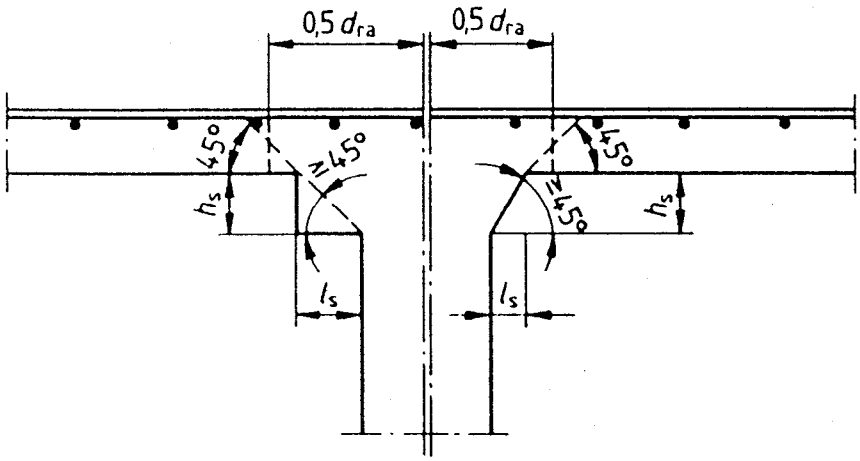
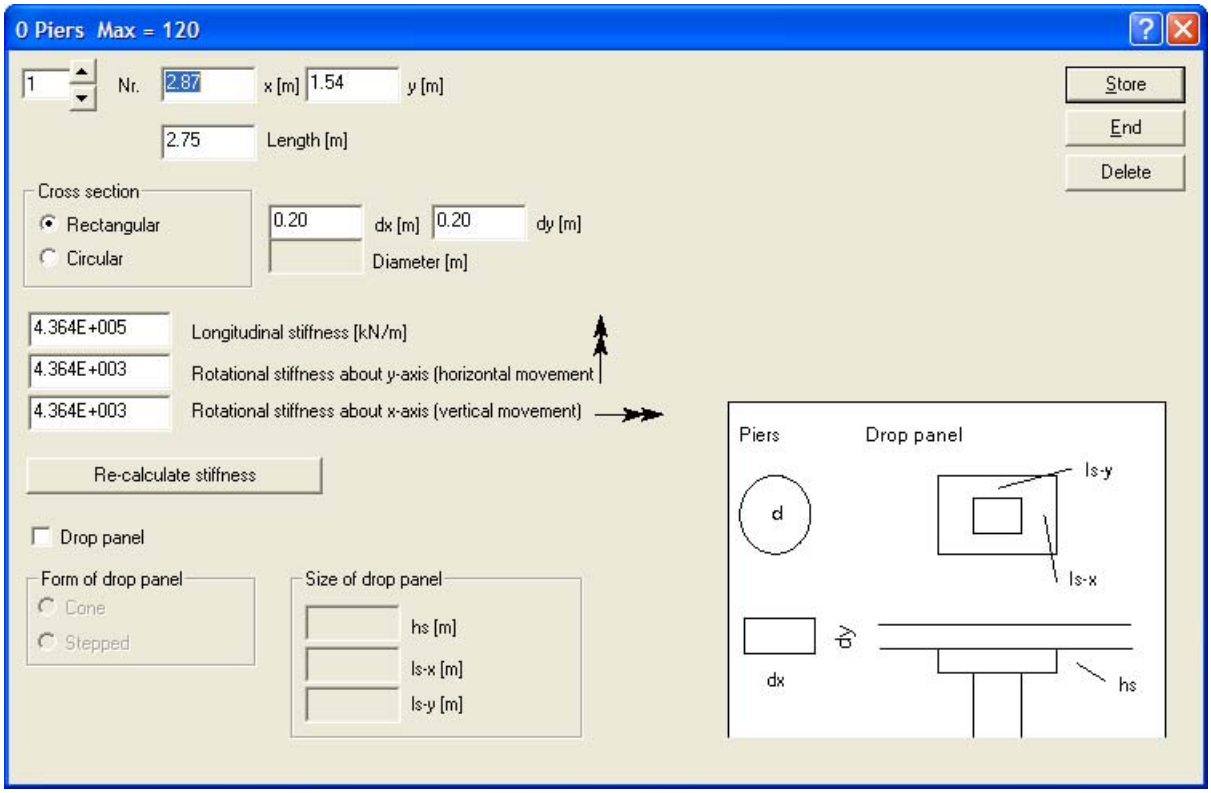


Fig. 3.26 Drop panels (left) and cone shaped (right) enlargements of the column head

3.17. Piers

In graphical input mode, you click on the icon

and next on the approximate location of the pier. In the dialog that pops up



you can specify the exact coordinates of the pier and additional parameters.

The additional weight of drop panels is considered automatically by the program.

Given the length l of the pier and the widths dx and dy the program calculates the longitudinal stiffness

$EA/l \qquad E = 3*10^7\ kN/m^2 \qquad A = area$

and the rotational stiffness in the x- and y-direction

$3\ EI_y/l \quad I_y = b_y b_x^3/12$ (rotation about the y-axis, corresponding to a movement in x-direction)

$3\ EI_x/l \quad I_x = b_x b_y^3/12$ (rotation about the x-axis, corresponding to a movement in y-direction)

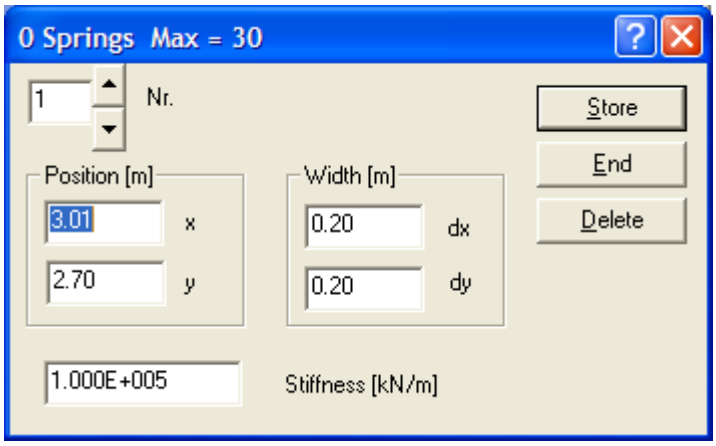
The factor 3 corresponds to a hinged support at the foot of the pier. For a clamped support, the factor 3 must be replaced by 4.

Of course, you can overwrite the default values.

3.18. Springs

In graphical input mode, you click on the icon

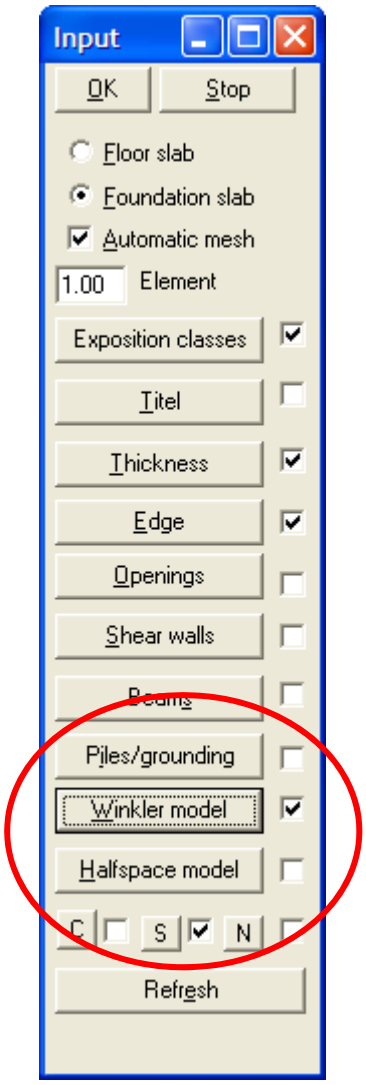
and next on the approximate location of the spring.



Unlike a pier a spring has no rotational stiffness.

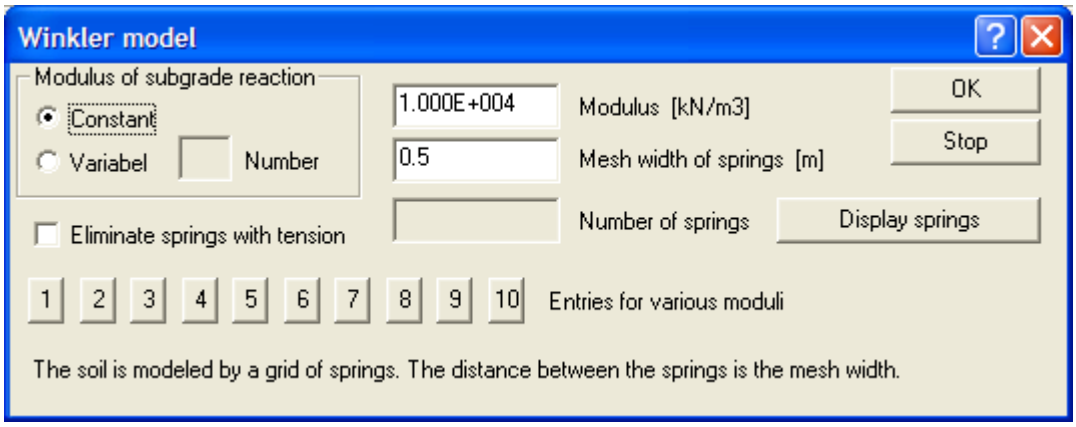
4. FOUNDATION PLATES

The analysis of foundation plates can either be based on the Winkler model or the half space model.



4.1. Winkler model

In the Winkler model, the soil is modeled by a series of single springs which can move independently of each other.



By checking the box

Eliminate springs with tension

the program can set the stiffness of such springs to zero. This is done in cycles. The program first finds these springs, eliminates these springs, and then restarts the analysis.

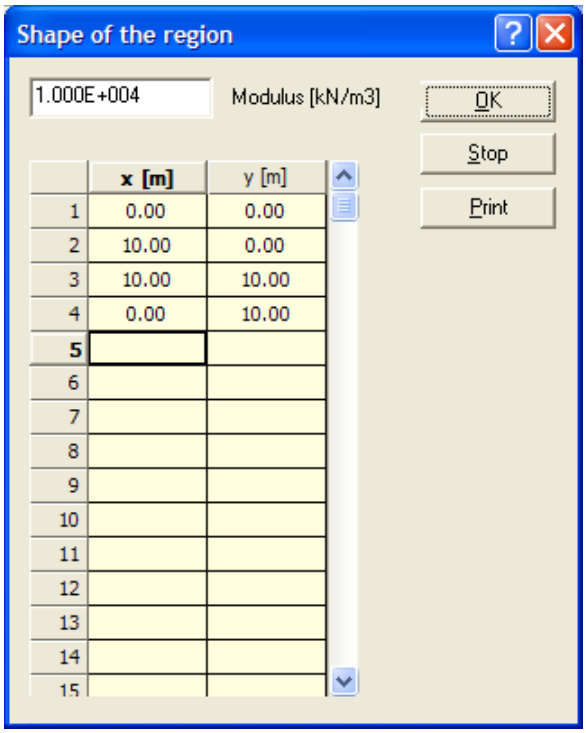
You may think twice before you activate this option. Firstly, the superposition of different load cases is no longer possible, and secondly, the analysis may not converge.

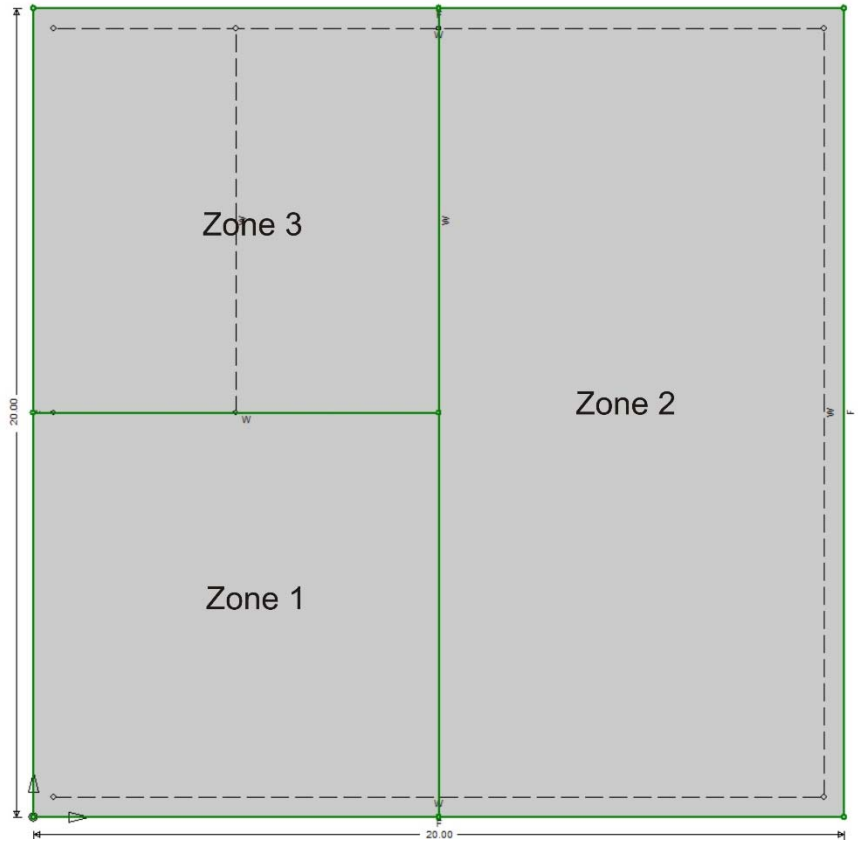
4.1.1. Modulus of subgrade reaction

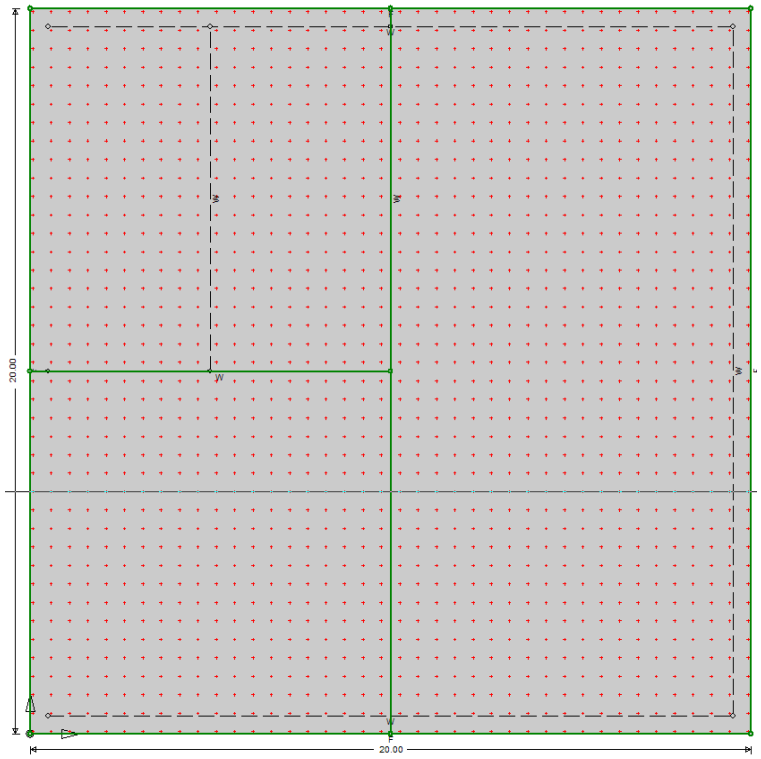
The modulus of subgrade reaction has dimension kN/m3 and lies in a range between 10.000 and 100.000 kN/m3.
The modulus can vary patch wise, that is the soil surface can be subdivided into up to six different zones or regions. The input of a single region is done by

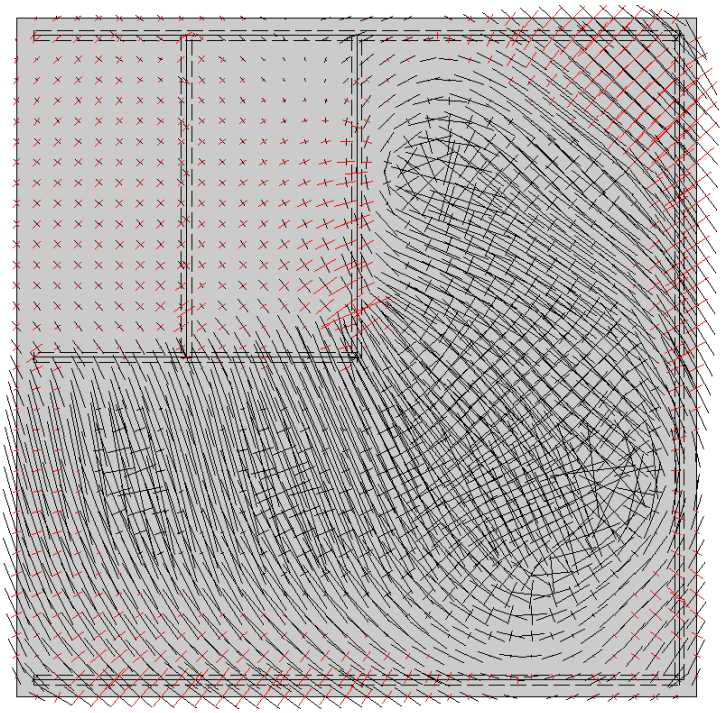
- **Specifying the number of vertices, the region has and**
- **the coordinates of the vertices**

The sense of rotation for a zone is the same as on the edge of the plate. This holds also true for zones which are embedded into other zones (while in panels which are embedded into other panels the sense of rotation is opposite to the outer edge of the plate).




Zone 3
Zone 2
Zone 1
20.00
20.00


20.00
20.00



4.1.2. Mesh width

In the Winkler model, the soil is represented by a grid of springs. The mesh width in x- and y-direction of this grid is specified by the user.
The stiffness of a single spring is therefore the product of the modulus c of sub grade reaction times the area of a single cell

Stiffness [kN/m] = c [kN/m3] * area of a cell [m2]

where the area of a single cell is

area [m^2] = mesh width x * mesh width y

4.1.3. Support conditions of the edge

Normally the edge of a foundation is a free edge (no restraints), but other support conditions such as rotational springs or vertical springs (in addition to the soil) are also admissible.

4.1.4. Shear walls

If you place a shear wall on the foundation plate the program assumes that you want to model the wall like an oversized T-beam with a very large bending stiffness EI.

Such shear walls should not be placed directly on the edge of the plate but they should be kept at a small distance, say 0.2 … 0.5 m, from the edge.

The loads that these walls carry can be input as line loads [kN/m].

4.1.5. Beams

Beams allow you to model the bending stiffness EI of these structural elements. So, if you want to have an influence of the stiffness of a shear wall, model the shear wall as a beam. Structurally they are equivalent.

4.1.6. Piles

By placing the plate additionally on piles, piled raft foundation can be modeled.

In such foundations, the piles as well as the raft transfer the building load. The piles transfer a part of the loads into deeper and stiffer layers of soil and so reduce the settlement of a building.

4.1.7. Foundations of piles

Where a column sits on a foundation plate the thickness of the plate is often increased to better accommodate the concentrated load.

To simplify the input of such a base plate, the base plate can be declared the drop panel of a (virtual) pier on the underside of the plate. Because only the drop panel is important and not the pier (which is non-existing), the pier is assigned a very small cross-sectional value and a very short length.

For the rectangular cross section of the imaginary pier choose dx = dy = 0.1 m and assign the pier a length of 0.1 m.

Assign the following values to the piers:

Longitudinal stiffness	= 1 kN/m	// cx
Rotational stiffness	= 0.0	// cphi_x, cphi_y

Specify a drop panel and design the drop panel according to the dimensions of the base plate of the column

ls-x	// Width x of the foundation = 2 * ls-x + dx
ls-y	// Width y of the foundation = 2 * ls-y + dy
hs	// hs + plate thickness = total thickness of the foundation

4.2. Half-space model

In the half-space model, the plate and the soil form a coupled problem. The deflection of the plate and the settlement of the half-space must be identical. This allows calculating the distribution of the soil pressure between the plate and the surface of the soil.
To model the soil pressure distribution the surface of the soil is divided into a mesh of constant pressure elements so that by checking the settlement at the mid-points of the elements the (approximate) distribution of the soil or pressure can be calculated.

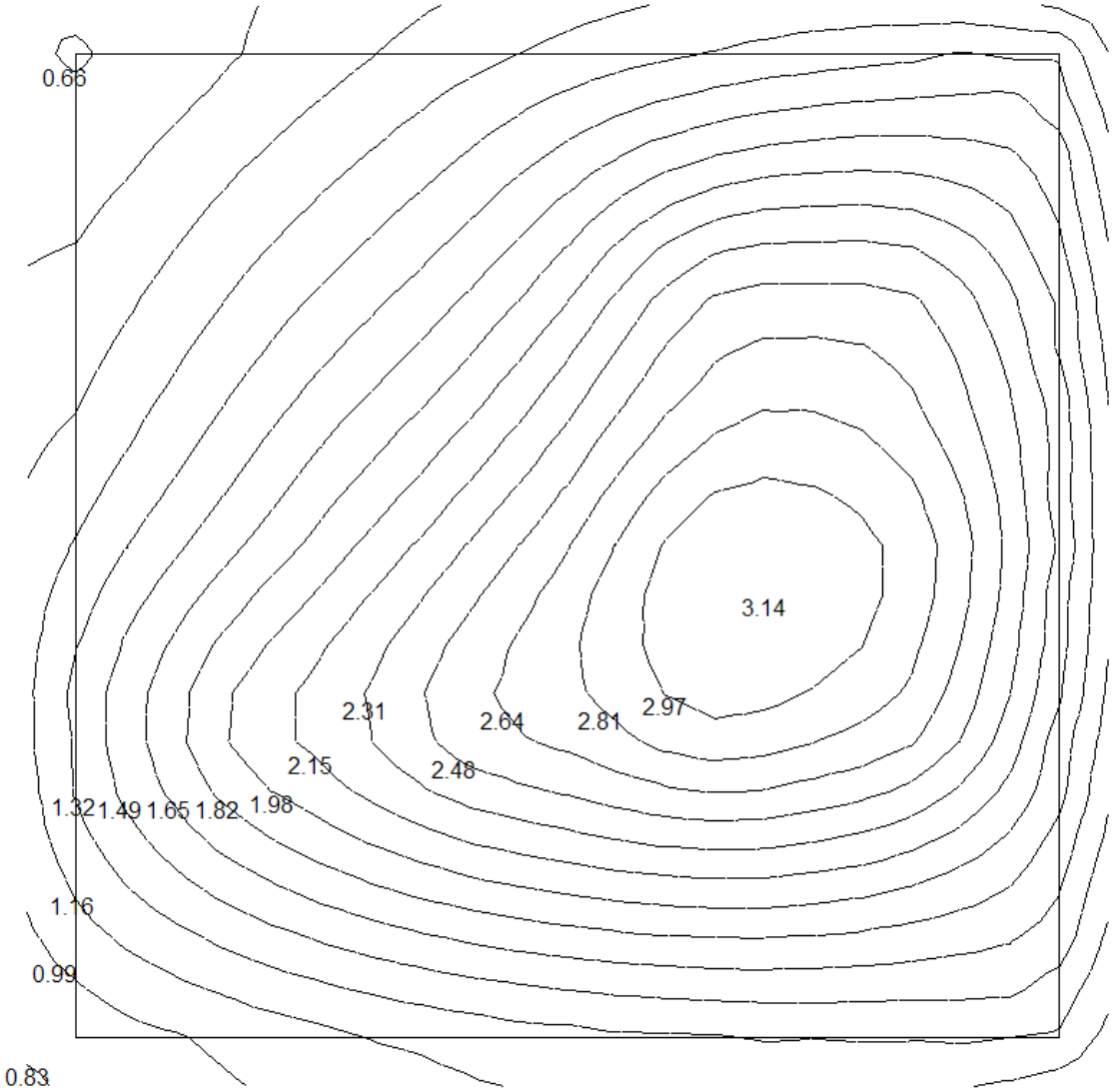
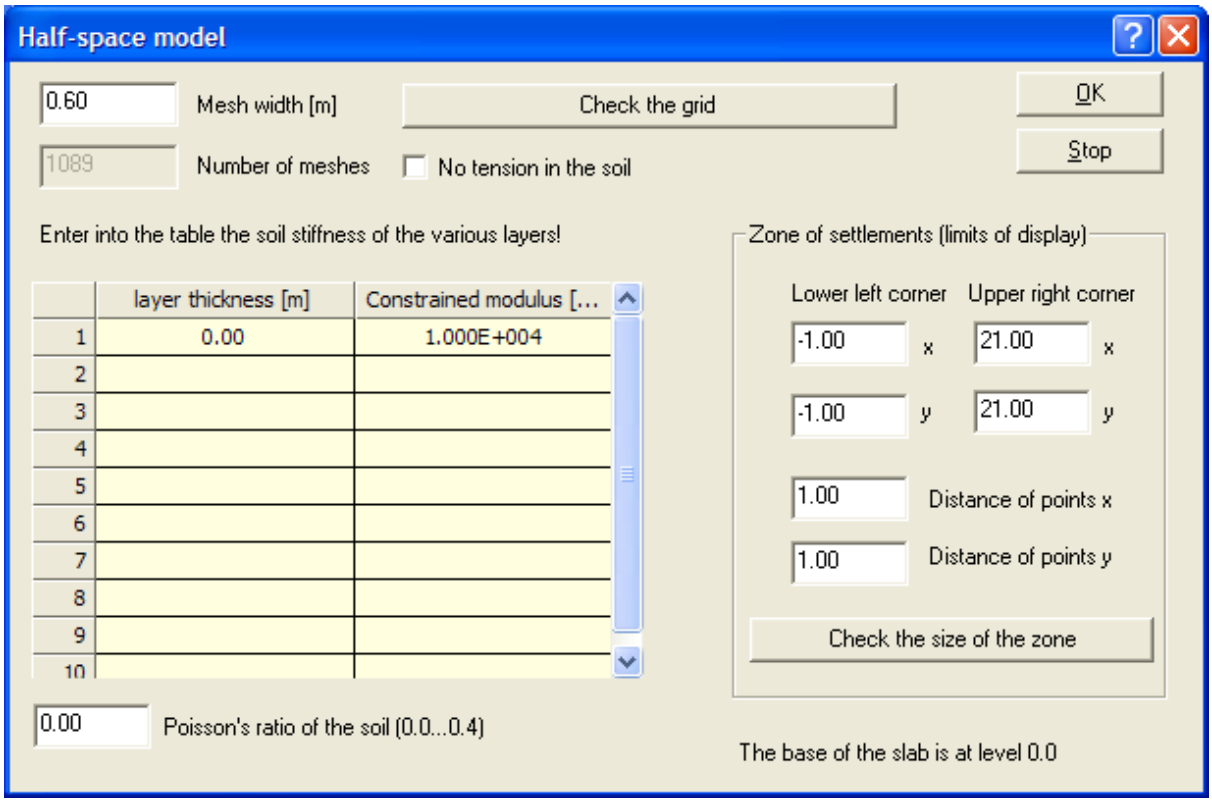


Fig. 4.1 Contour lines of the vertical displacement of the soil surface



4.2.1. The soil

The elastic properties of the soil are given in terms of the *constrained modulus of elasticity* of the soil and Poisson's ratio.

The soil can consist of up to ten different layers. Each layer is specified by

- the vertical coordinate y of the top of the layer (at the soil surface y = 0)
- the constrained modulus of elasticity of the layer
- Poisson's ratio

The first layer starts at y = 0.

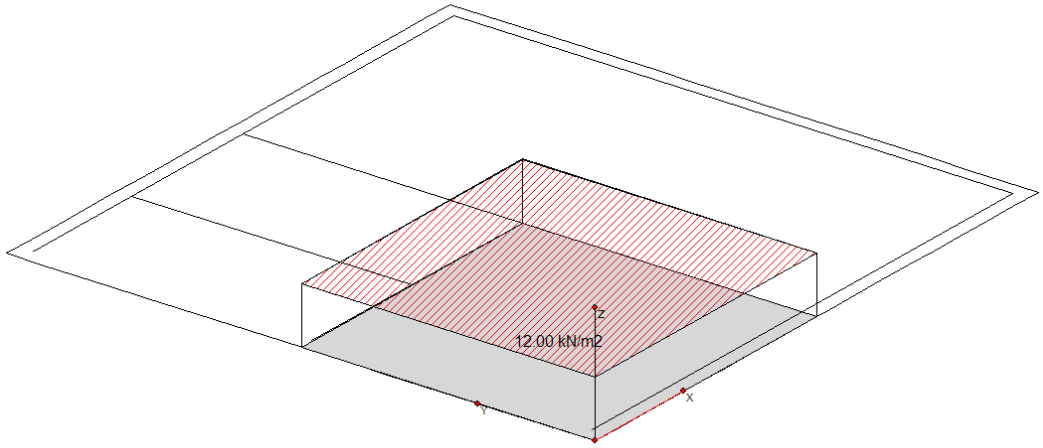
4.2.2. Mesh width

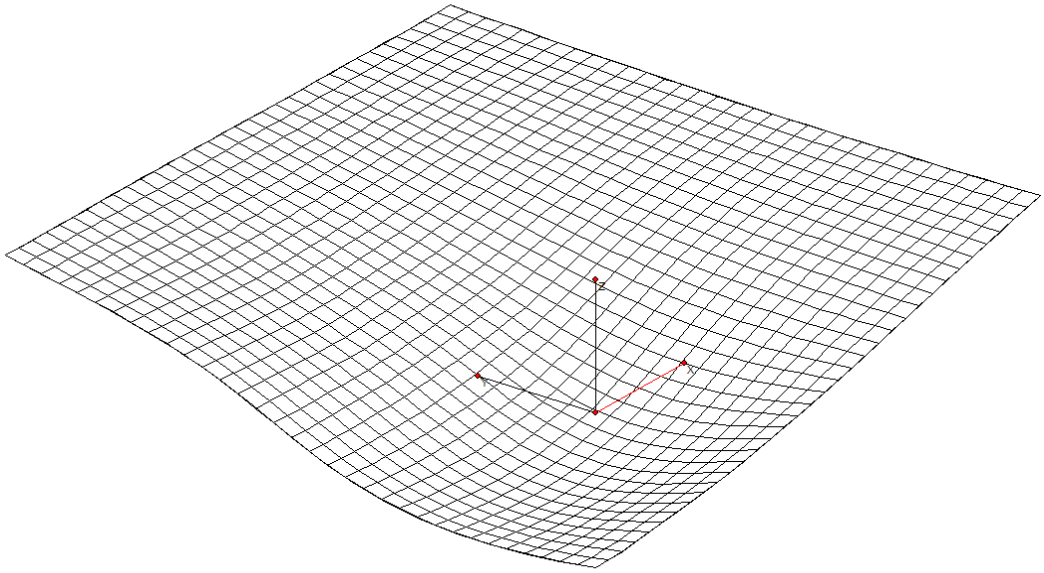
The mesh for the soil pressure distribution consists of quadratic elements d x d so that you only need to specify the value d, the mesh width.
The smallest value of d possible is determined by the maximum number of cells the mesh can have.

4.2.3. Settlements

In addition to the soil pressure distribution the program also calculates the settlement of the elastic half-space in the vicinity of the foundation plate.

How far this region extends into the surrounding area is specified by the user. In any case this region has a rectangular shape so it suffices to specify the endpoints of the first diagonal. But in addition, you must specify the mesh width in x- and y-direction of the grid that is to cover this region. The program calculates the settlement at the nodes of this invisible mesh. Do not choose too small a mesh size because the storage is limited at this point. A mesh width of 1 m in each direction seems reasonable.

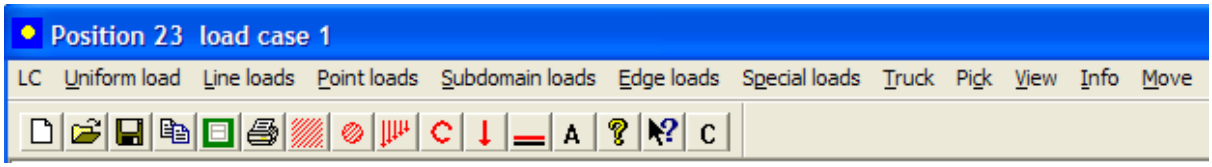

12.00 kN/m2
z
x
y


z
x
y

5. LOAD

Possible loads are

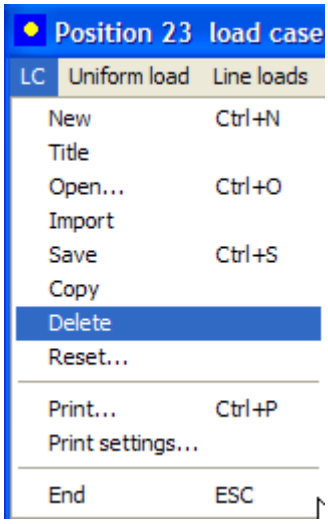
- **gravity load**
- **uniformly distributed load**
- **partial area loads**
- **line loads**
- **line moments**
- **single forces, single moments**
- **edge moments, edge forces**
- **settlements of piers and walls**
- **temperature changes**
- **dislocations and rotations at single points**
- **truck loads**
- **influence surfaces**



To enter a new load case, click on the icon

To store a load case, click on the icon

You can also import or copy load cases from other positions



5.1. Uniformly distributed load

This can either be a constant load p or the dead load g of the plate.

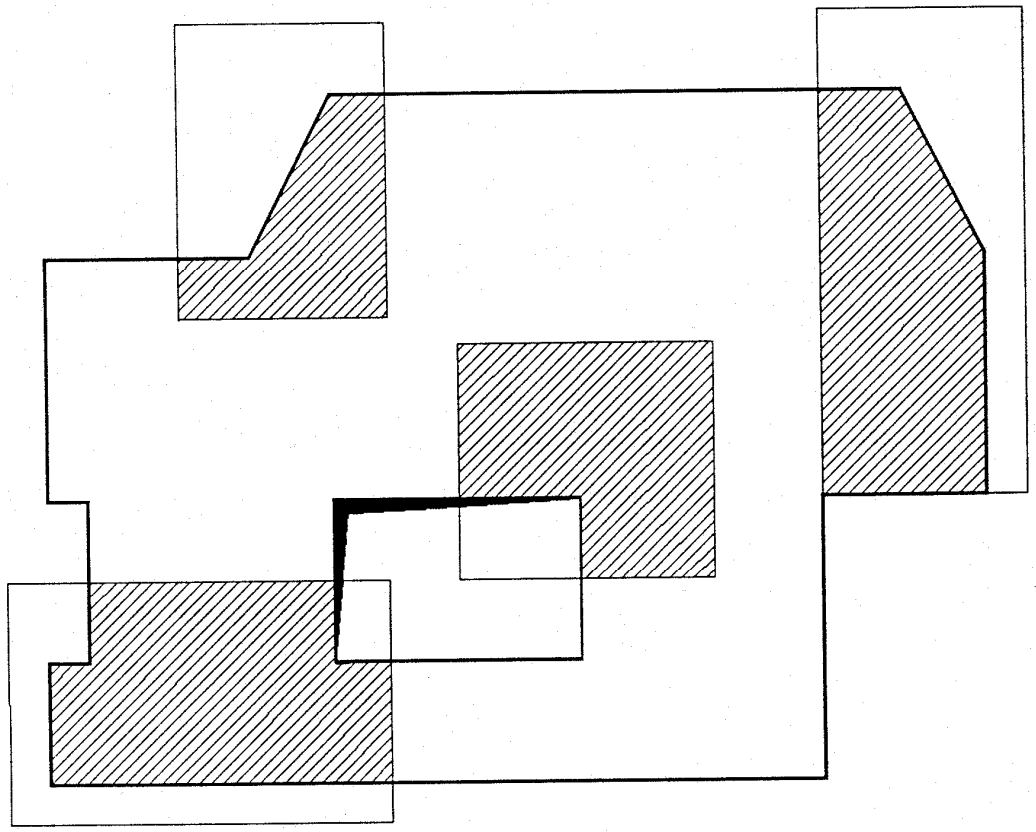


Fig. 5.1 Generating partial area loads with rectangular covers

The dead load of the plate depends on the specific weight of the concrete (kN/m^3) plus a surcharge (kN/m^2) for the pavement or a detached ceiling.
The program automatically considers the thickness of the different panels, the additional weight of drop panels etc.

5.2. Partial area loads

The partial area can have any shape. To start the input, click on the icon

and next on the corner points of the partial area. The sense of rotation is the same as on the outer edge. To close the partial area, return to the starting point for you can also press the S-key. Next enter the magnitude of the load. Do this also if the load is not constant.

If the load is not constant, next click on the menu entry **Partial area loads**, to modify your input by specifying the values of the load at the vertices of the partial area.

In this tabular input mode, you can also make corrections to the coordinates of the vertices.

If the partial area is a rectangle then it suffices to specify the endpoints of the diagonal. To this end first click on the lower left corner of the sub domain, next on the upper right corner of the partial area and finally press the right mouse button

- lower left
- upper right
- right mouse button

to enter the magnitude of the load.

You can also specify the shape of the partial area by covering the plate with a rectangle. That part of the plate which is covered by the rectangle is the partial area, s. Fig. 5.2.

To avoid ambiguities, extend the cover well beyond the plate that is do not stop at a corner point of the plate but move outside of the plate.

Linear partial area loads such as

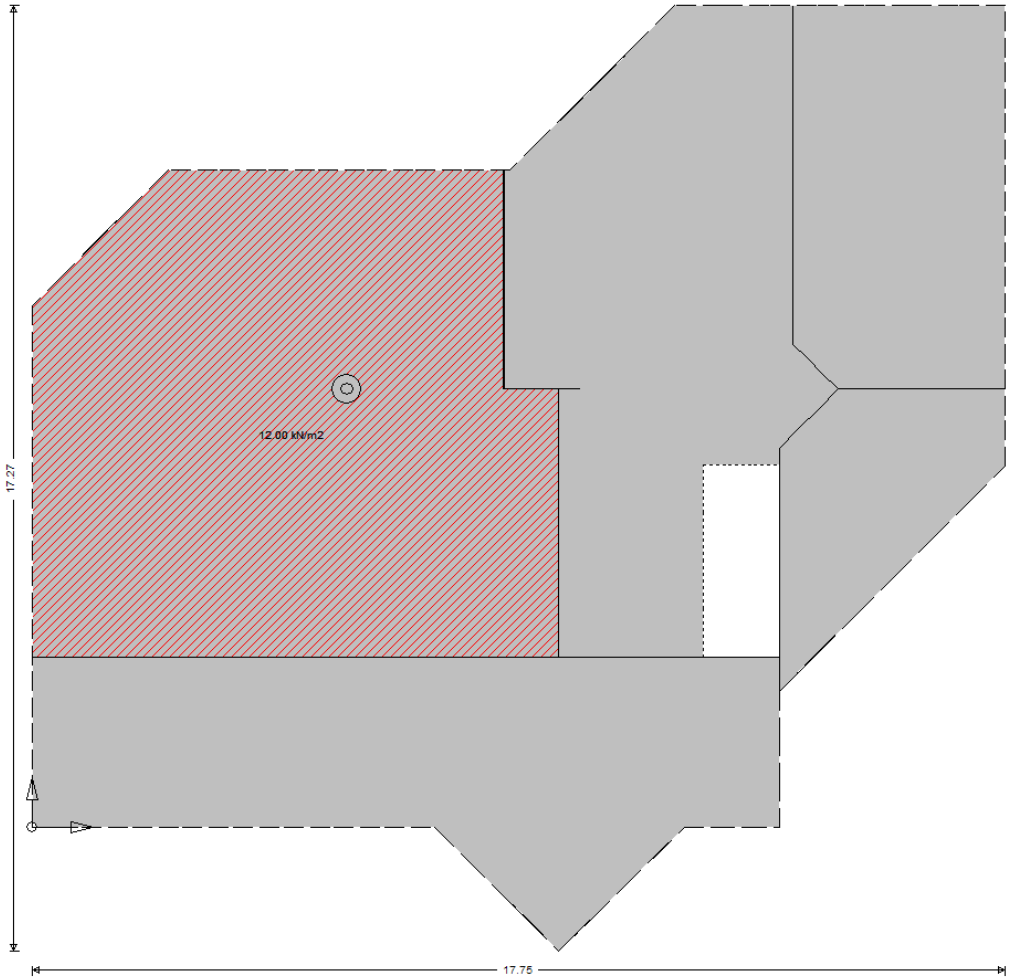
$$p(x,y) = a + b * x + c * y$$

can be modeled exactly by the program.

In all other cases (quadratic or cubic variations) the program will approximate the loading by doing a first order Taylor expansion that is it will approximate the load p by an inclined plane.

To check the approximation the program outputs the exact and the interpolated values of the load at the corner points of the partial area.

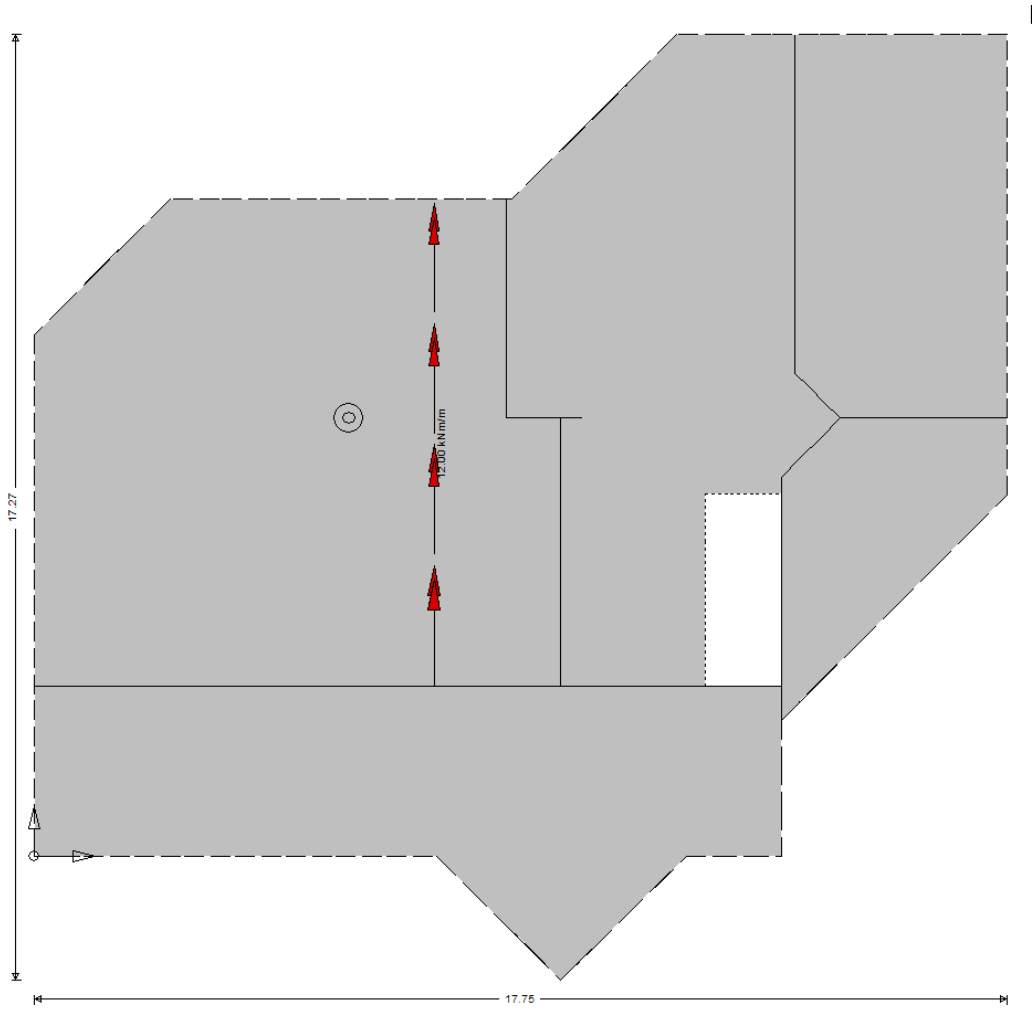
If an opening of the plate lies inside a rectangular cover the program will automatically subtract the opening from the partial area.



5.3. Line loads

Line loads can be vertical forces [kN/m] or moments [kNm/m]. Line loads which act along the edge of the plate can also be entered as edge loads.

To start the input click on the icon



and then on the endpoints of the line load. In the context menu that opens you can specify the type of load and the magnitude of the load. When you enter additional loads the context menu will stay shut, because the program assumes that the next loads are of the same type and magnitude. To activate the context menu, click the right mouse button.

5.4. Point loads

Point loads can be of static type

forces
moments

or of ‚geometric' type, that is:

a bent	(will generate the influence surface for bending moments)
a twist	(will generate the influence surface for the twisting moment m_{xy})
a dislocation	(will generate the influence surfaces for shear forces)

Point forces look the same from all directions but all other point loads act in a certain direction in the x-y plane. For example, a single moment **M** = (M_x, M_y) rotates about an axis and so it 'rolls' in the direction orthogonal to the rotation axis. This direction is specified by the unit vector

n = [n_x,n_y] (length of the vector = 1)

where

$n_x = \cos\alpha = M_x / M \qquad n_y = \sin\alpha = M_y / M \qquad M = (M_x + M_y)^{1/2}$

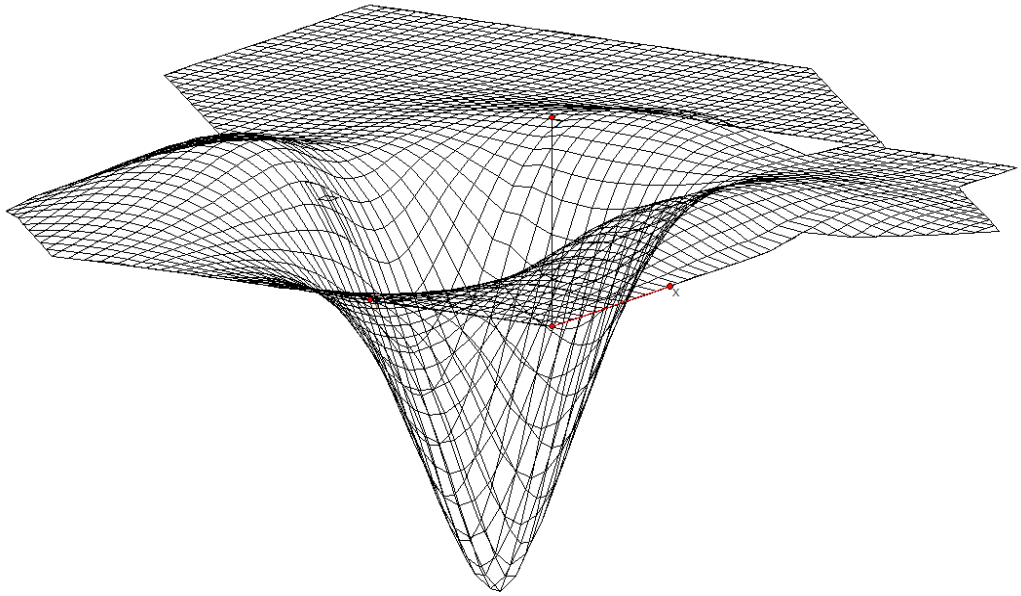


Fig. 5.4 Influence function for bending moment m_{xx} at a certain point in a plate

Simply said the vector **n** indicates the direction of the main curvature produced by the moment vector **M**. A moment **M** = (M_x, 0) (curvature in x-direction) has the direction **n** = [1,0] and a moment **M** = (0, My) (curvature in y-direction) the direction **n** = [0,1], s. Fig. 5.3.

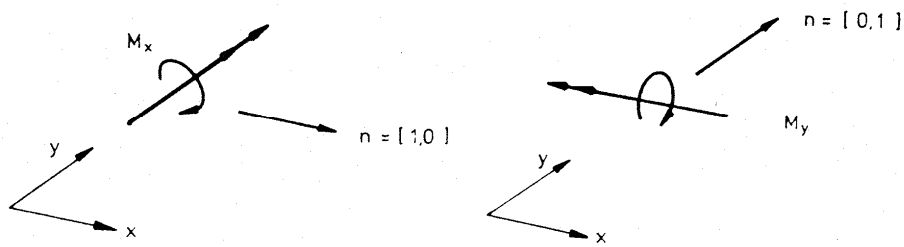


Fig 5.3 Single moments **M** = (M_x, 0) and **M** = (0, My)
To generate influence surfaces for one of the following quantities at a point x, y you must apply the following point loads:

Influence surface for	type of point load	n_x, n_y
w	force	/, /
$w_{,x}$	moment	1, 0
$w_{,y}$	moment	0, 1
m_x	bent	1, 0
m_y	bent	0, 1
m_{xy}	twist	1, 0
q_x	dislocation	1, 0
q_y	dislocation	0, 1

All point loads are of magnitude 1. The influence surface is the deflection surface of the plate under the action of these point loads. To start the input, click on the icon

and next on the approximate location of the point. In the context menu, you can specify the exact coordinates of the point and the action of the point load.

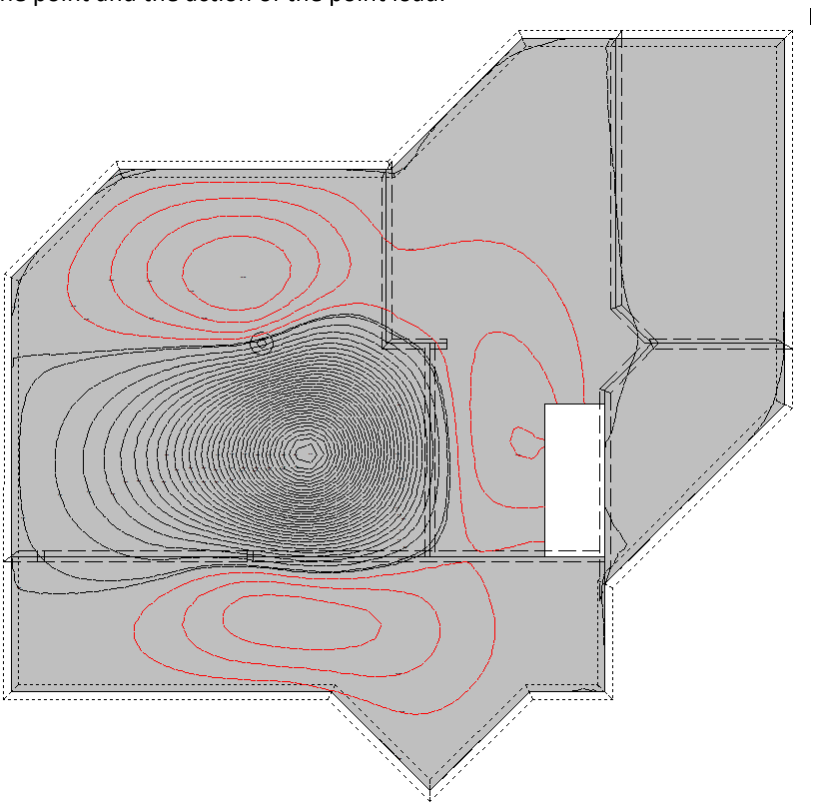


Fig. 5.5 Contour lines of the influence function

5.5. Edge loads

Edge loads can be line loads [kN/m] or moments [kNm/m]. They act along single sides of the plate.

What is the difference between edge loads and line loads? None - from a practical point of view. But internally edge loads are treated like inhomogeneous boundary values while line loads appear on the right-hand side of the differential equation.

Mathematically it is a sounder approach to model line loads which act directly along the edge of the plate as edge loads.

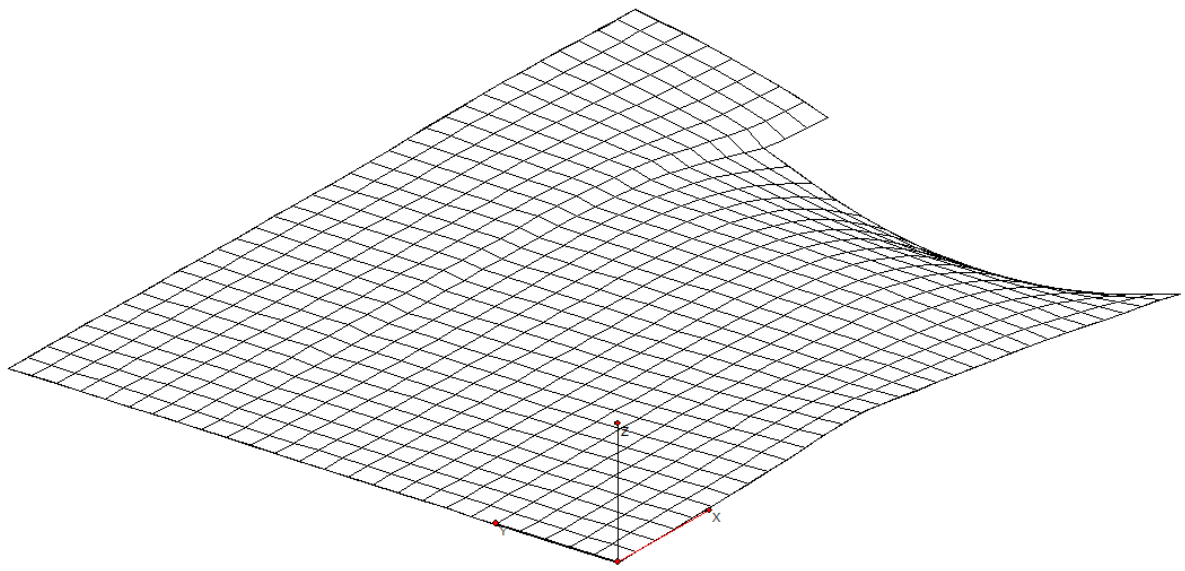


Fig. 5.5 Effect of an edge load

Remark 1: Edge loads which push the edge down are positive but in they appear as negative edge forces v_n (Kirchhoff shear) in the output because they pull the edge down.

Remark 2: The program does not accept vertical edge loads along hinged or otherwise fixed edges. If you want to place a line load directly along such an edge, enter the load as a line load.

Because of these two remarks we would recommend that you enter vertical edge loads as line loads, but line moments on edges should be entered as edge loads.

To start the input of edge loads, click on the icon

and next on the corresponding edge.

5.6. Settlements

Piers and walls can settle. To start click on

Special loads -> Settlements

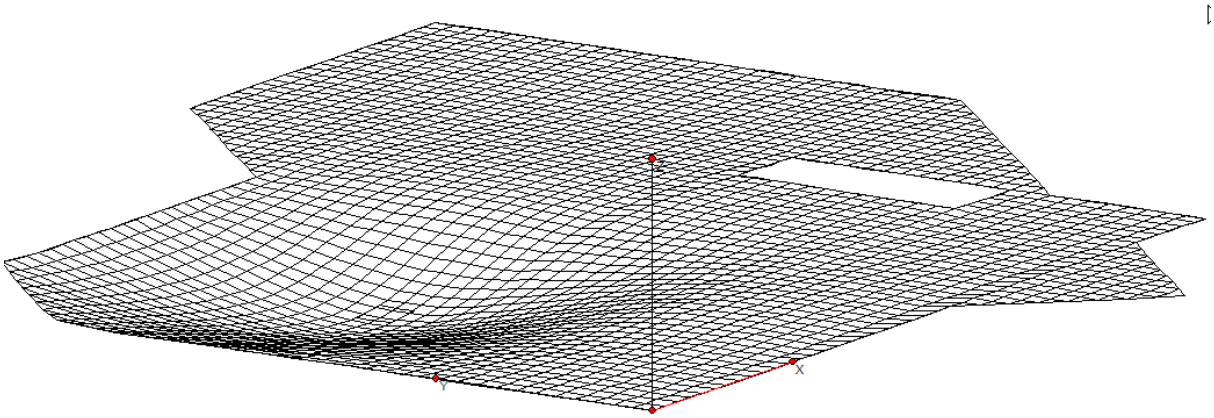


Fig. 5.6 Settlement of a pier

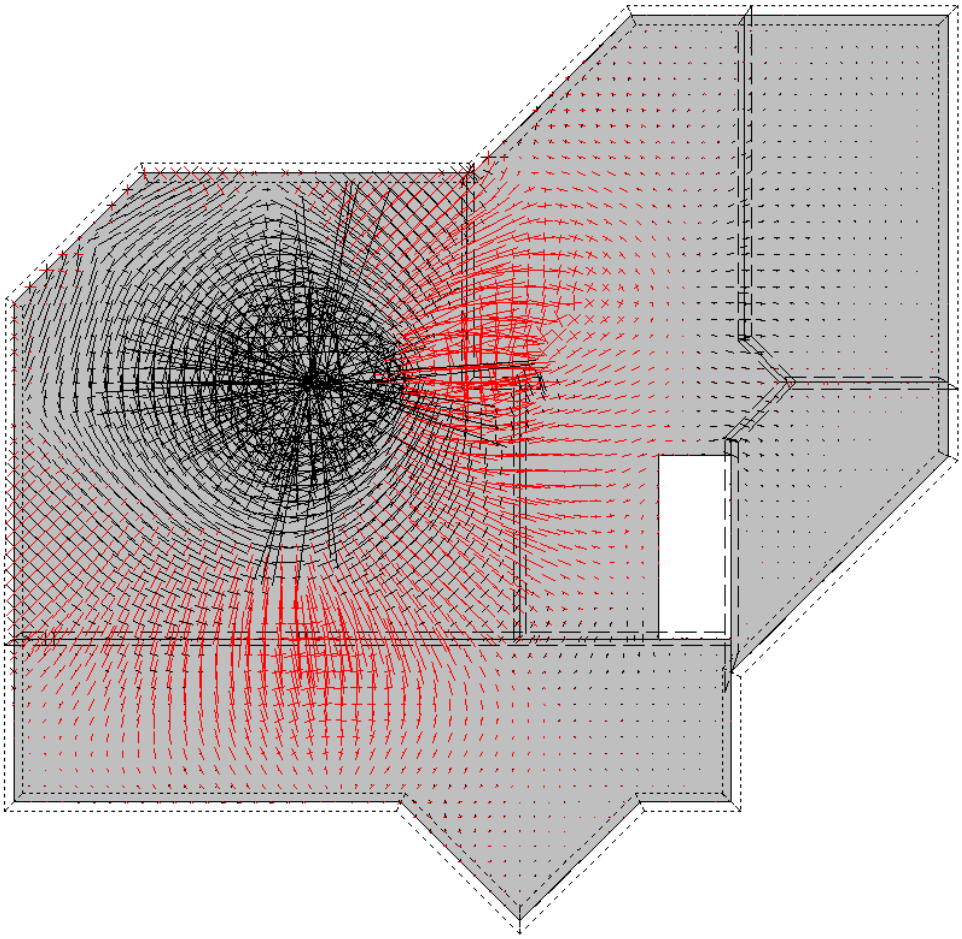


Fig. 5.7 Principal moments resulting from the settlement

5.7. Temperature

To consider effects due to changes in the temperature you need to input the temperature coefficient α_t = 1.0 1.2 E-5 of the material and the temperature difference ΔT between the upper and the lower side of the plate. A positive difference means that the lower side is warmer than the upper side.

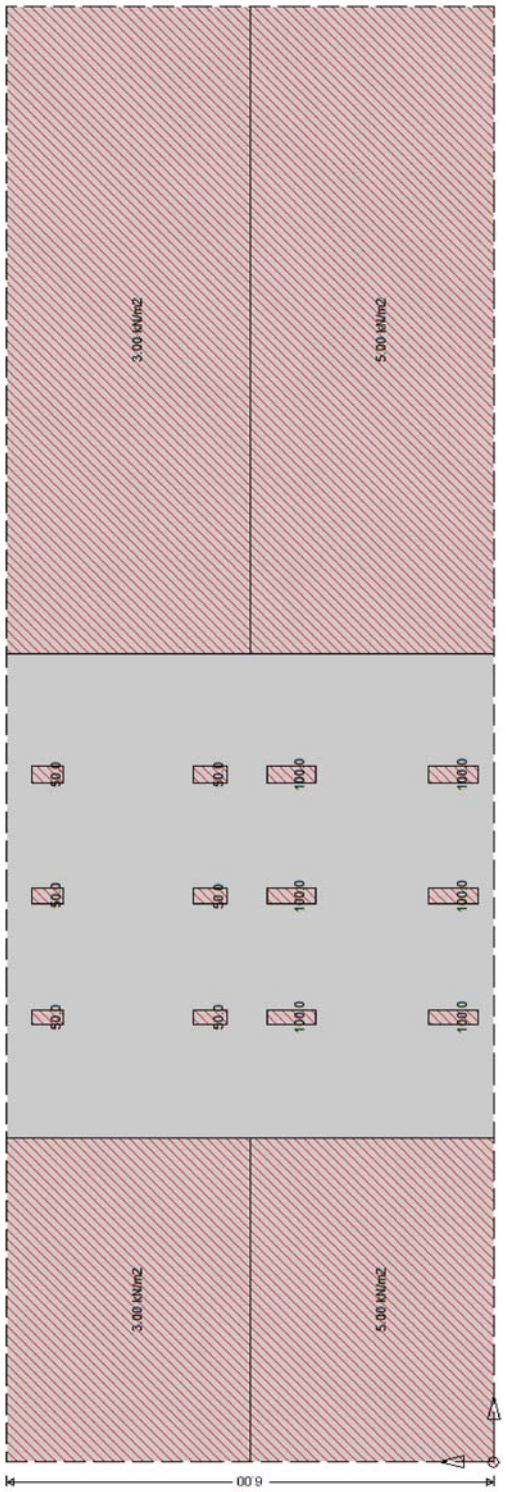
To start click on

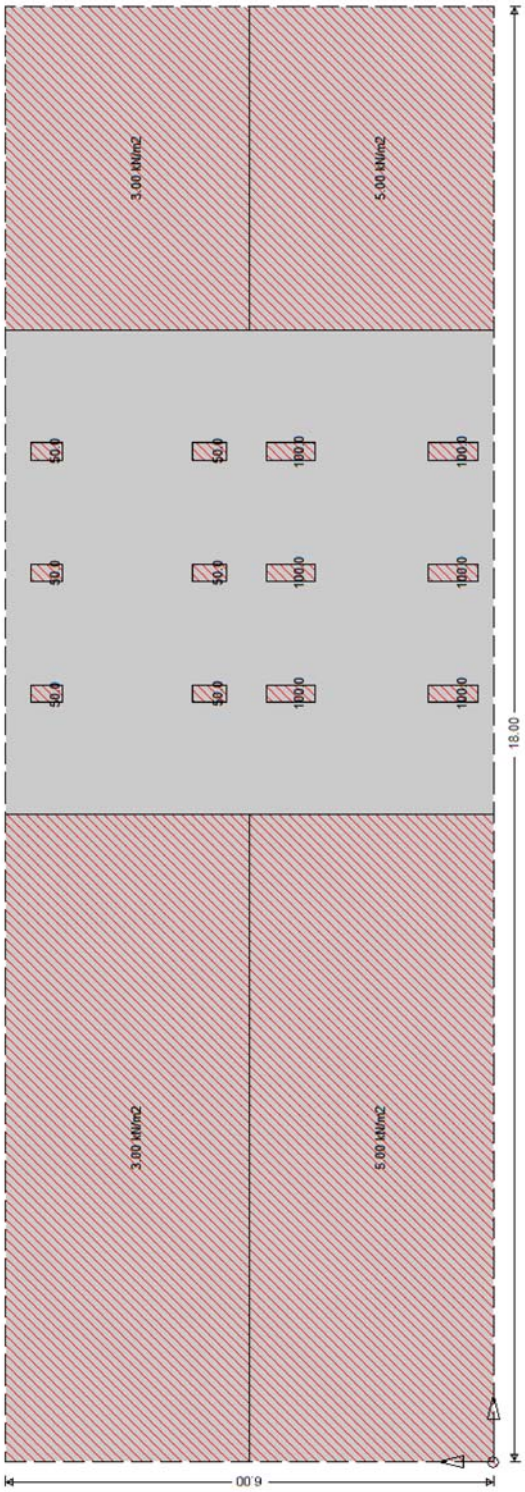
Special loads -> Temperature

5.8. Truck loads (bridge classes 60/30 and 30/30)

The program can generate a series of load cases to simulate the crossing of a bridge by a truck. Click on the menu entry **Truck** and follow the directions.

Each lane is specified by the position of its four corner points. Next enter the vibration coefficient, the weight of the truck, and the distance between the parking positions of the truck.


3.00 kN/m2
5.00 kN/m2
50.0
100.0
6.00


3.00 kN/m2
5.00 kN/m2
50.0
100.0
6.00
18.00

6. Stress points

The stress points are the points for which stresses and deformations are calculated. The

stress points (BEM)

are, so to speak, the

nodes and Gauss points (FEM)

of the boundary element method.

Because in the boundary element method there are no nodes in the interior of the plate these points must be generated separately. But one can place the stress points freely across the plate. The accuracy does not depend on the size of the cells of the grid, not on the distance between the single points. Each point is independent from its neighbor. The stresses are not based on an interpolation but each stress value is calculated by a proper influence function which mainly 'lives' on the boundary.

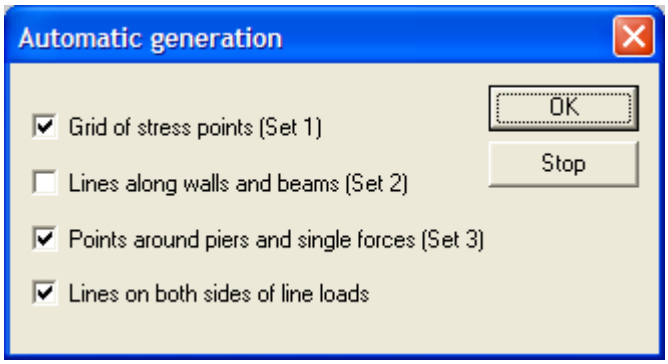
For example, you might place a stress point directly on a wall to calculate the bending moment at this point. Now the program automatically calculates also the shear force at this point. But the shear force is discontinuous across a support and you cannot expect that the influence function for the shear force (which is a very precise mathematical tool) can handle such a situation. The result for the shear force is unpredictable – and rightly so. But if you move away from the support by, say 10 cm, the shear force is again well defined and the influence function will function properly.

The stress points can form a complete grid or a partial grid, they can form straight lines, they can form clusters of points, for example clusters of four points at each pier, or they can be isolated single points with no apparent recognizable pattern in their arrangement.

Stress points come in **sets**. That is, you can have different arrangements of stress points and not just one mesh as in the finite element method.

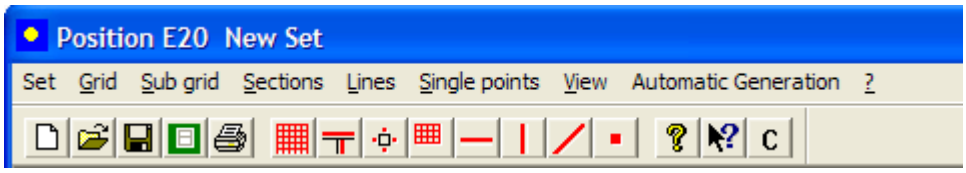
6.1. Let the program do the work

Depending on what kind of plate you have the program will generate the default sets of stress points automatically for you - if you want.



But you can do it also yourself, of course.

6.2. New set



To enter a new set of stress points, click on the icon

and to store the set – after it has been generated - click on

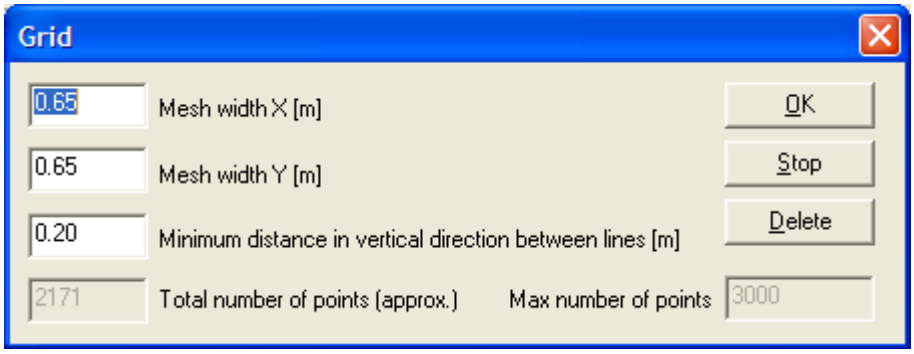
6.3. Grid

By clicking on the icon

you can generate a grid of points which covers the whole plate or by clicking on the icon

you can generate a sub-grid, which only covers a part of the plate.

The grid (mesh) is determined by the mesh width in horizontal and vertical direction. The grid is not a mesh but rather a set of horizontal lines on which the points are distributed in (approximately) equal distances.



The distance of the lines in the vertical direction is controlled by the mesh width in vertical direction.

The program will try to align the lines of the grid near openings in such a way that the lines will come in contact with the lower or upper edge of an opening (because we want to know the stresses at the edge of an opening). If the edges of two openings – sitting side by side - are almost on the same level then this would force the grid to have two horizontal stress lines with nearly zero vertical distance. To avoid this, you can assign a minimum distance in vertical direction to the stress lines.

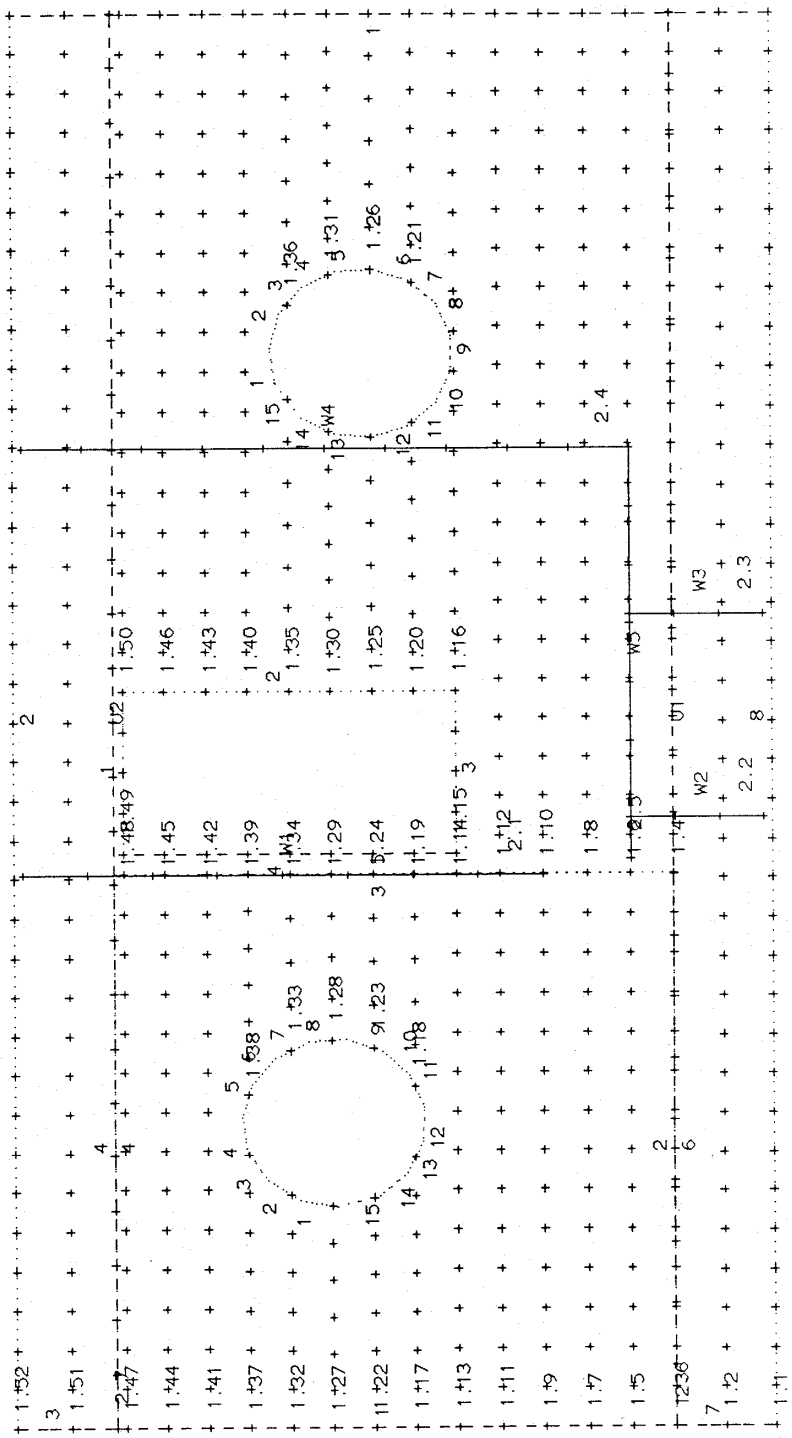
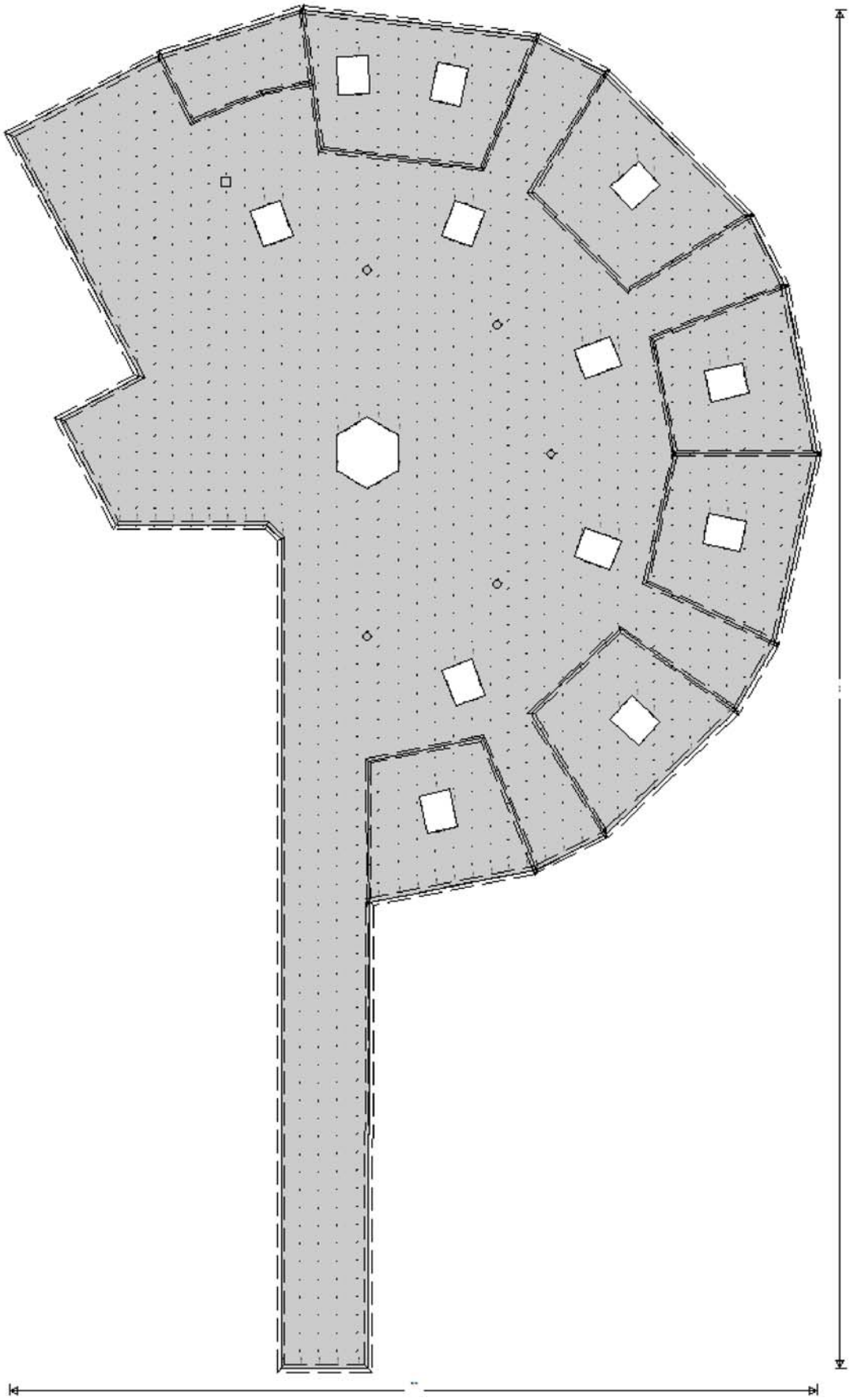
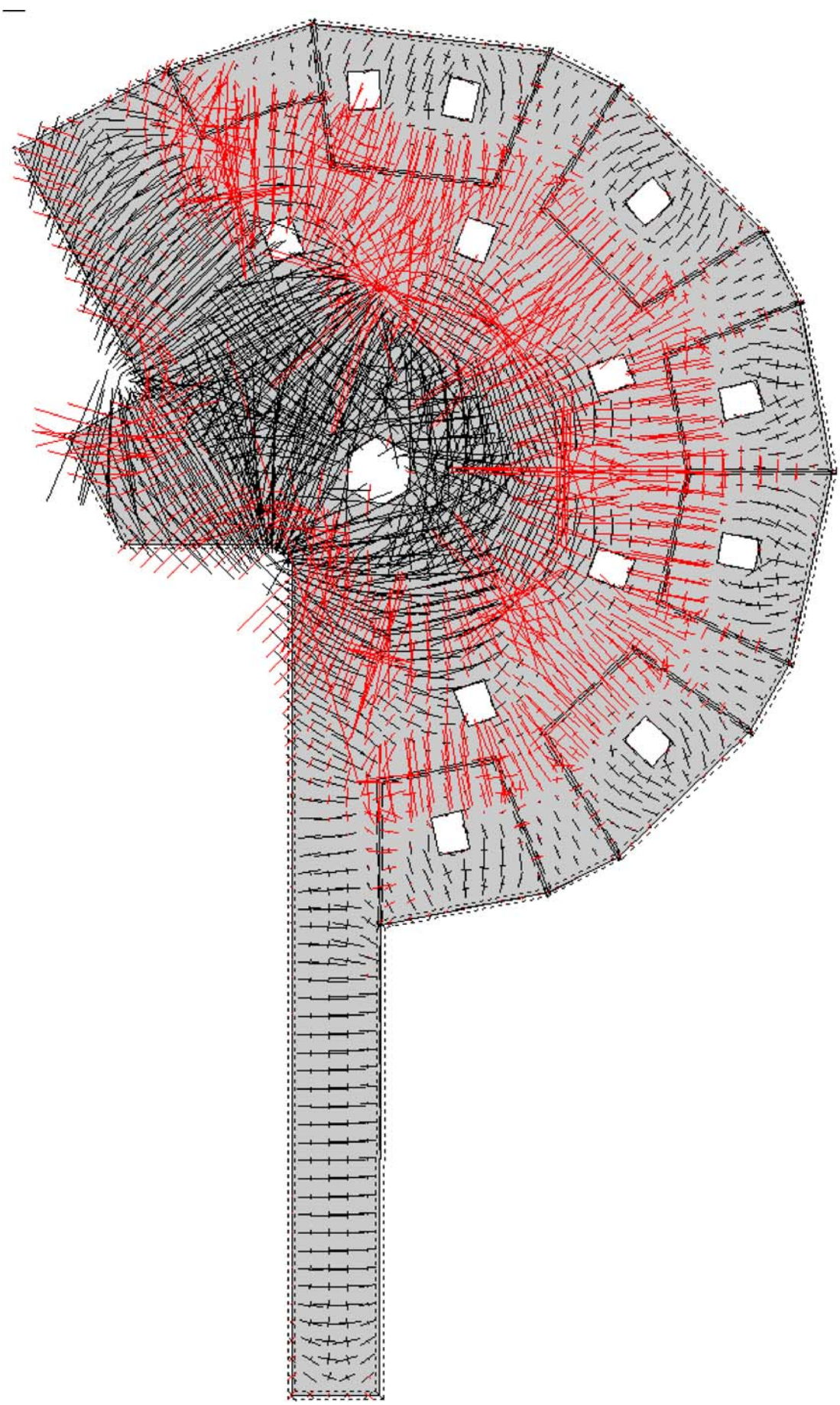


Fig. 6.1 A grid of stress points





6.4. Single lines

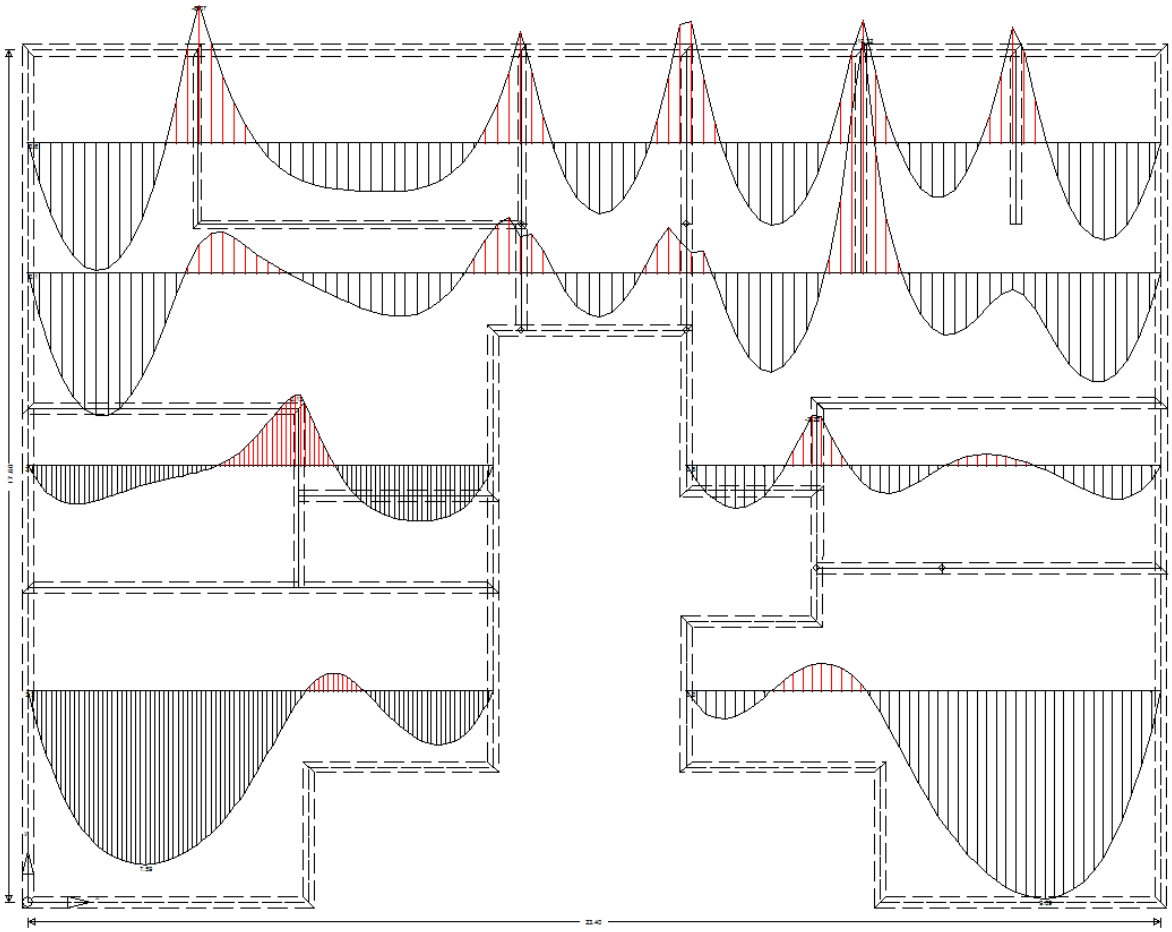
You can distribute stress points along lines which run in horizontal or vertical direction through the plate.

horizontal direction

vertical and direction

or you can have lines of stress points which start at a point A and end at a point B.

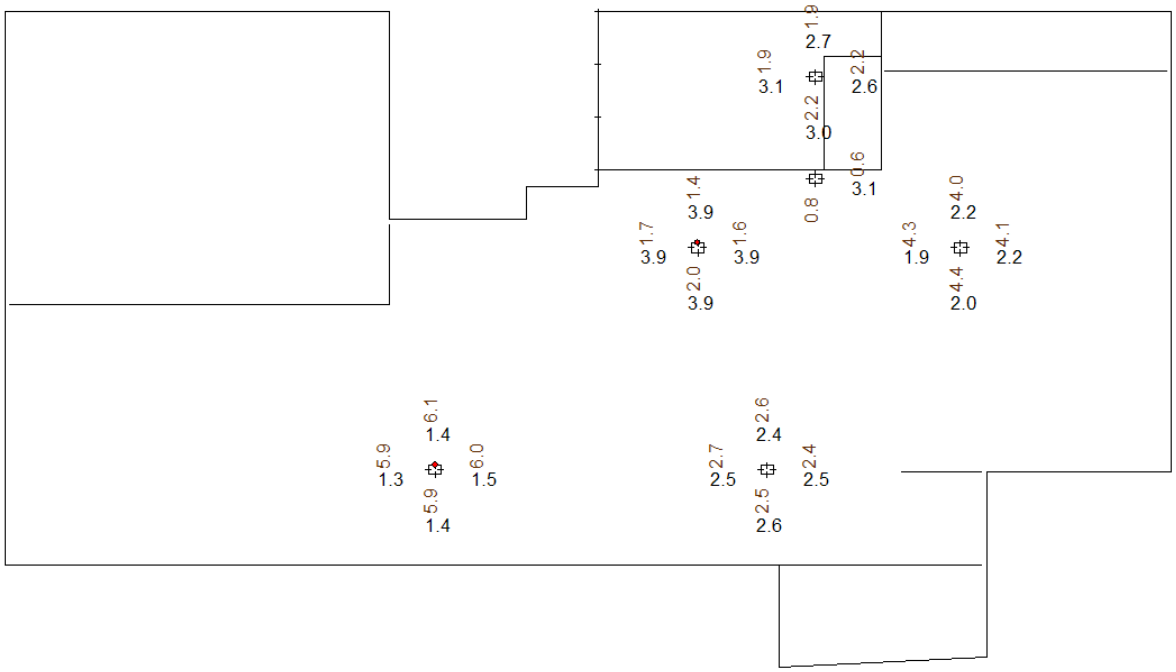
Lines which intersect openings or which run across different panels are split into separate parts.



6.5. Single points

You can also specify single stress points by clicking first on the icon

and next on the location of these single points.



6.6. Moments at supports and near point loads

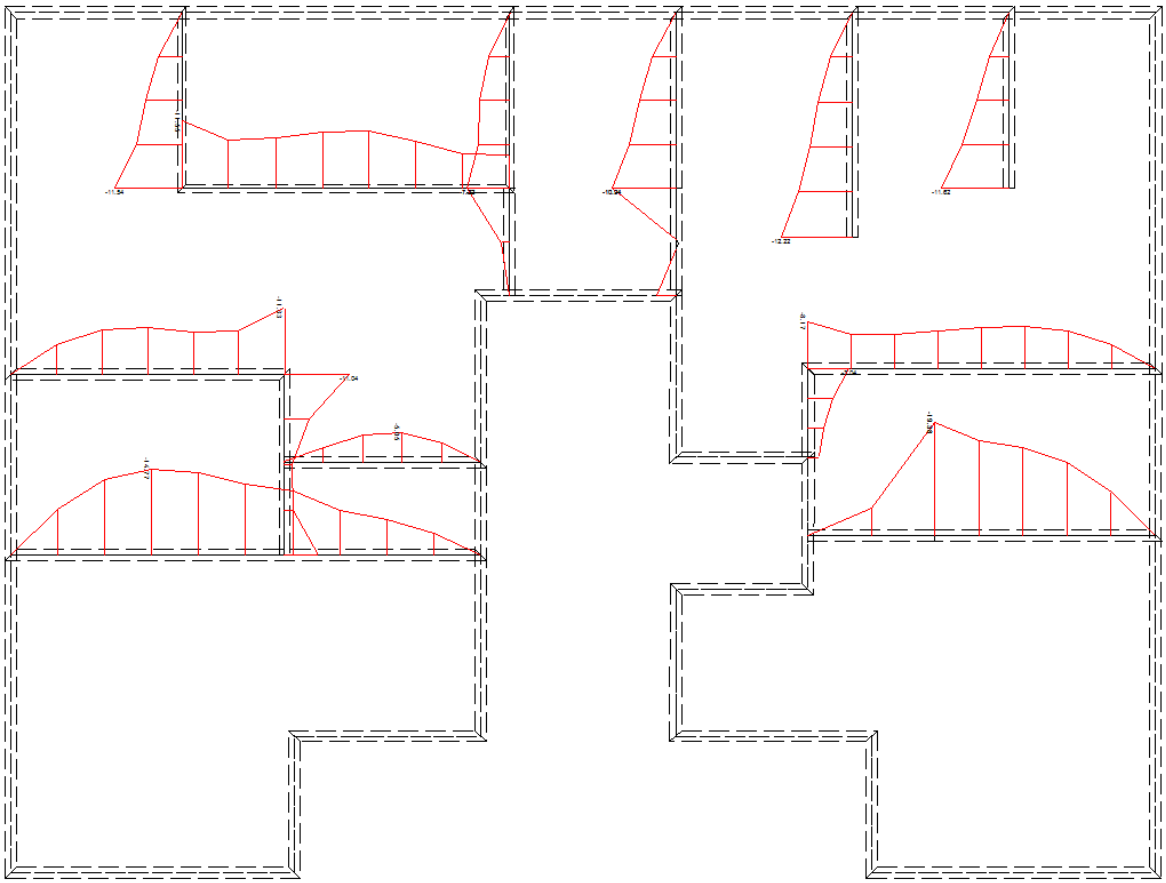
Near point supports and near point loads the gradient of the bending moments is large and therefore a tight control of such points is necessary. To this end the program generates at each pier or foot of a point load a cluster of four points (N – E – S – W) which encircle the critical point and so facilitate the analysis of the bending moment distribution near such points.

To activate this option click on the icon

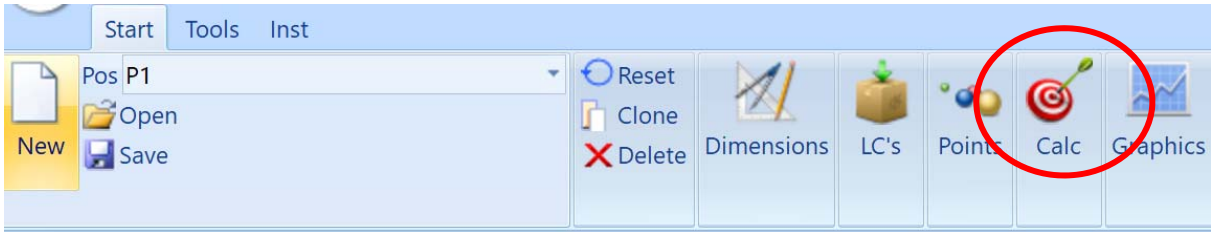
6.7. Moments along line supports (walls and T-beams)

By clicking on the icon

the user can generate stress lines which run along the axes of walls and T-beams. These lines better allow to study the bending moment distribution across line supports.

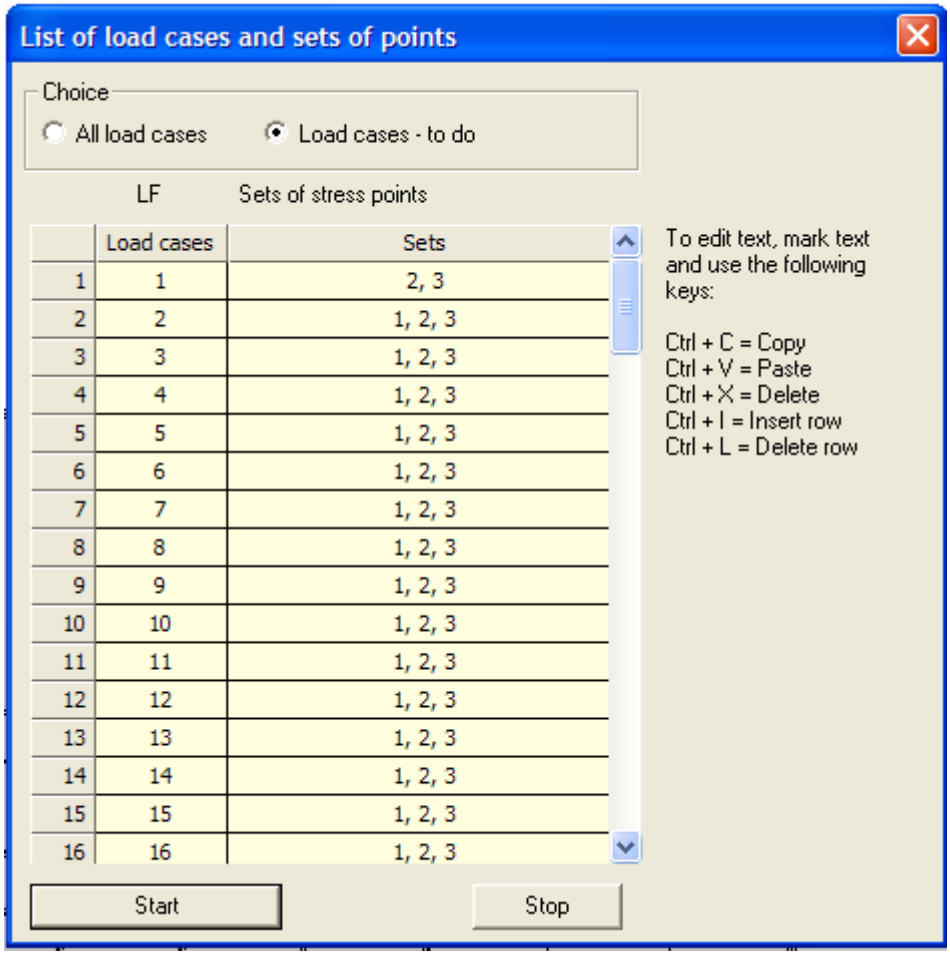


7. START OF ANALYSIS



After the input of the plate, the load you start the analysis of the plate by clicking on the button **Calc** in the main menu.

7.1. Start



In this table are listed the load cases the program will analyze and the sets of stress points for which stresses and displacements are calculated. This table can be edited.

7.2. Modifications

After an analysis of the plate you can, of course, add new load cases or generate new sets of stress points

- **additional load cases**

- **additional sets**

and let the program analyze these additional problems.

- **Modifications of the plate itself**

After any modifications of the plate, that is a change in the thickness or a change in the support conditions or if you add an opening to the plate you must call the discretization of the plate anew.

With any call to the discretization, this is the icon

in the part of the program where you input the dimensions of the plate, the plate is reset, that is any results of a previous analysis is considered no longer valid and the program expects that you restart the analysis after a call to the discretization.

You can check this behavior: Do an analysis of a plate, simply call the discretization of the plate anew, and turn on the graphical display of the results. All the menu entries which allow you to see the support reactions, the stresses in the plate etc., we'll be grayed out. To see again results you must start the analysis again.

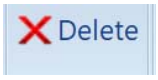
You can achieve the same result if you click on the icon

This also will reset the position.

There might be also situations where it is helpful to **clone** a position.

To duplicate means to make a copy of the actual position with all the load cases and all sets of points.

While resetting a position or duplicating a position leaves the position intact a click on the icon



will delete the position from the disk.

If you want to save your input before you delete any files click on the icon

8. RESULTS

8.1. Positive and negative forces

Support reactions are positive. Forces that pull the edge down are negative.

8.2. Equilibrium

In a boundary element program, the sum of the support reactions must **not** be equal to the sum of the applied load. It is **not** an error. But the difference between the applied load and the sum of the support reactions is normally very small, say 1%. If the error is larger than 3 % to 5 % the element length should be decreased. If this has no effects the discretisation itself should be checked.

Remark: In the finite element method, as in the boundary element method the programs solve an approximate load case exactly and the support reactions maintain equilibrium about the loads of these approximate load cases. So, a discrepancy of 1% signals that the total load in the approximate load case differs by 1% from the total load of the original load case.

You have the same situation in the finite element method. It is only that the finite element method better hides its error. In the boundary element method, the support reactions are the 'real', the physical support directions of dimension kN/m. And the sum of the support reactions is the integral over these distributed forces.

In the finite element method, you'll get 'nodal forces', that is the work done by the real support reactions on acting through the nodal unit displacements. So, the real support reactions are wrapped up, are weighted with the unit displacements. And these 'pseudo forces' (of dimension kN m = work) are output as support reactions – which they are not in the true sense.

For more on this topic see: F. Hartmann, C. Katz, Structural Analysis with Finite Elements, 2nd Ed., Springer-Verlag, p. 184

8.3. Support reactions

The support reactions have dimension kN/m and fixed end moments have dimension kNm/m.

The support reactions (and all other quantities as well) are influenced by the elastic properties of the supports. So, care should be taken to model these properties correctly.

8.4. Internal actions

Bending moments

m_x require reinforcement in x-direction and bending moments

m_y require reinforcement in y direction

So, in the same sense shear forces

q_x or q_y

(eventually) require shear reinforcement in x- and y-direction.

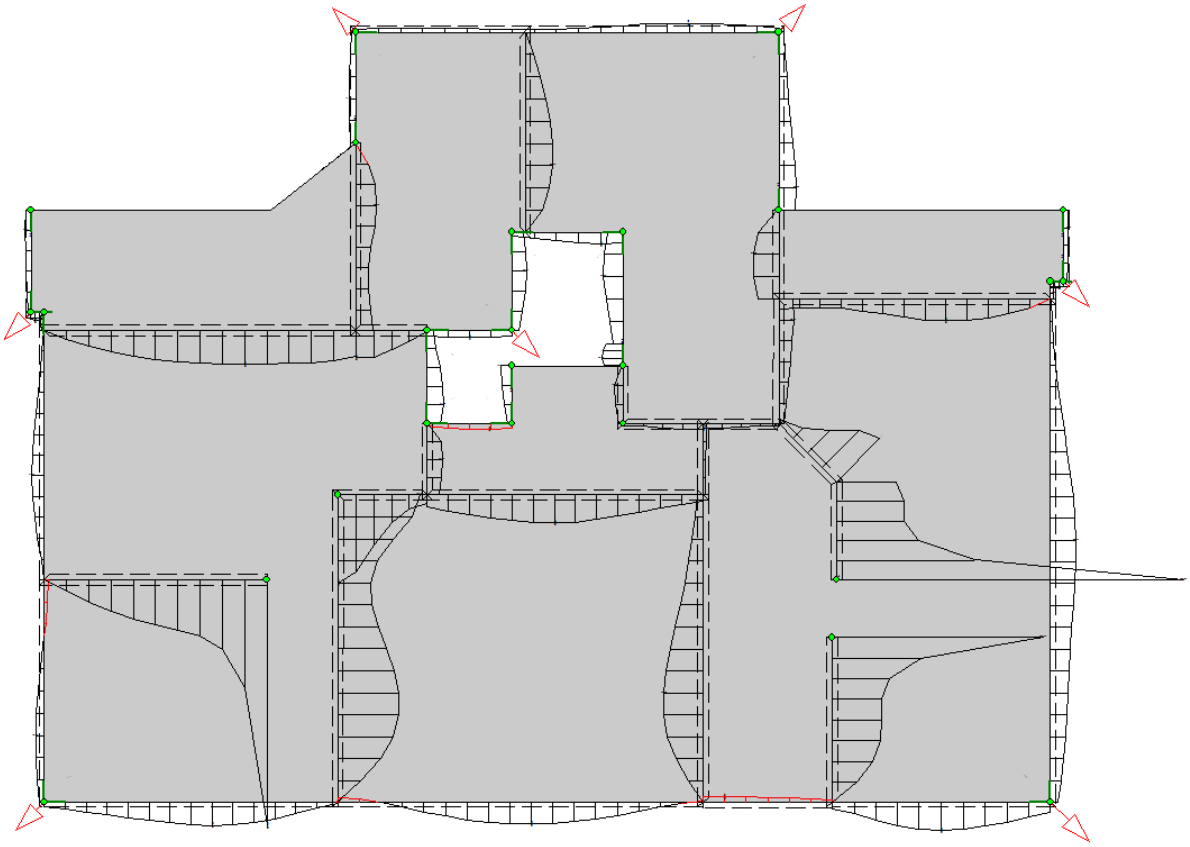


Fig. 8.2 Support reactions in a plate, kN/m

8.5. Boundary values

The program outputs the following boundary values:

w	deflection
w_n	slope
w_t	the slope in tangential direction (parallel to the edge)
m_n	fixed end moment
m_t	the bending moment parallel to the edge
m_{nt}	twisting moment
v_n	Kirchhoff shear (= support reaction)
q_n	shear force in normal direction

Add to this the (possibly) a single force F at corner points.

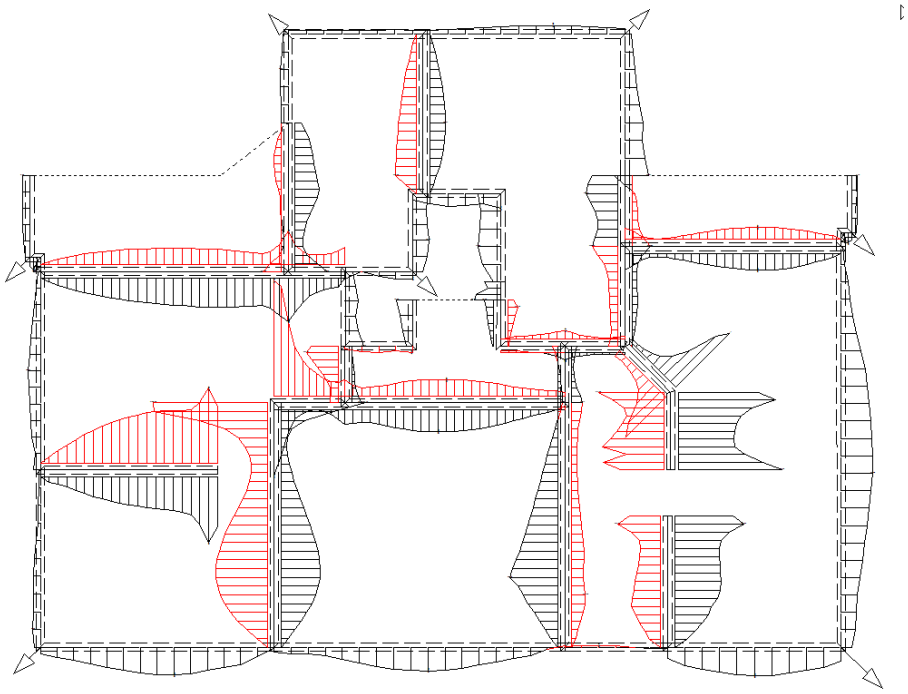


Fig. 8.2 Shear forces near the supports

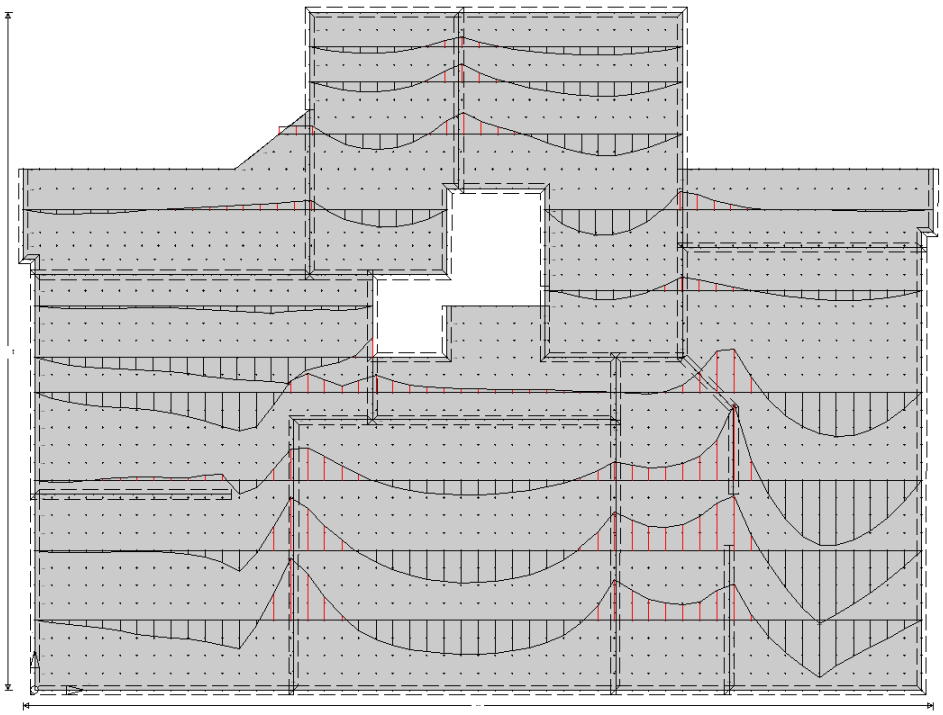


Fig. 8.3. Moments m_{xx} along some stress lines

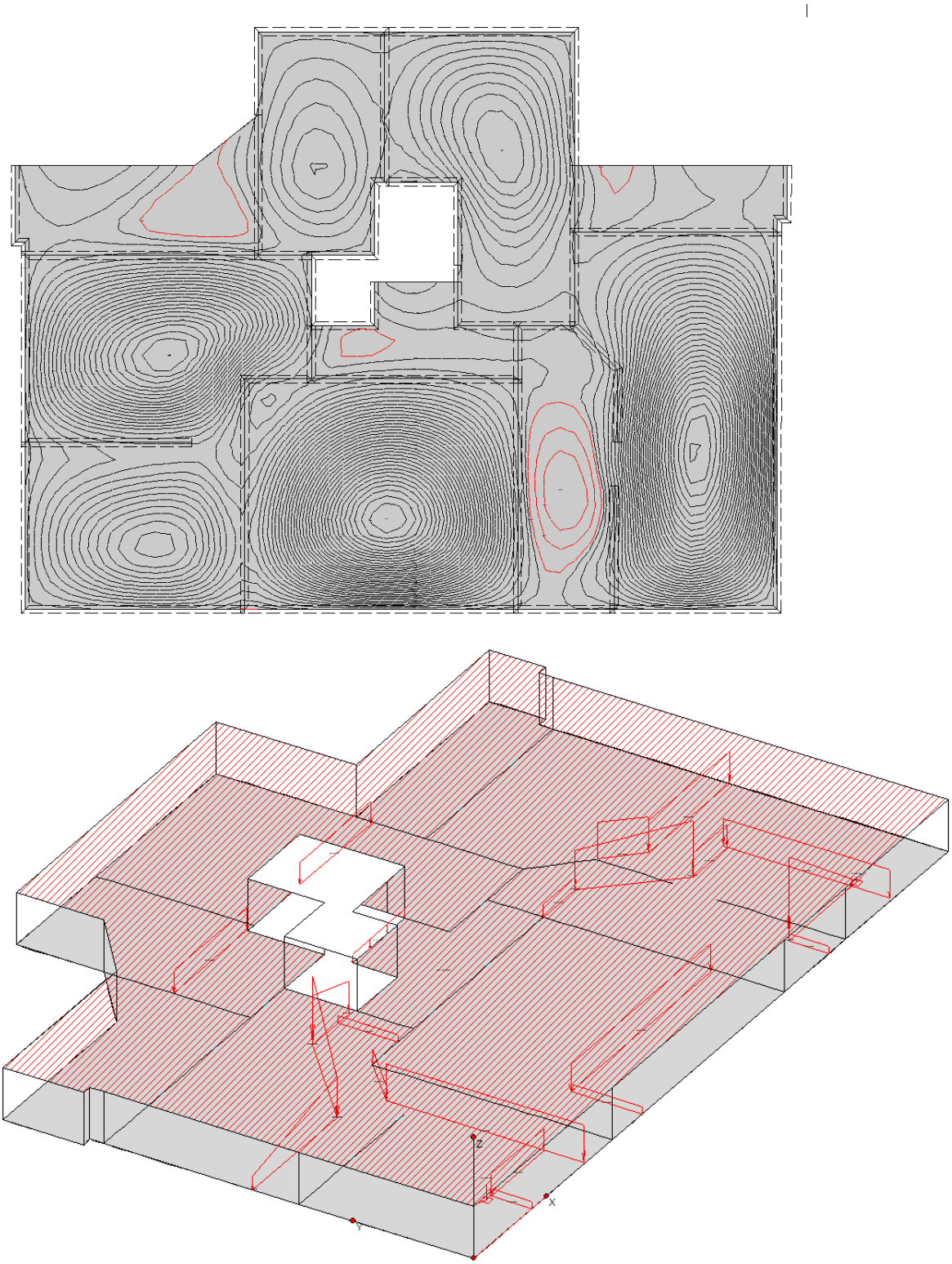


Fig. 8.4 Contour lines and 3-D plot of the load

8.6. Shear forces near walls and T-beams

Across a wall the shear force q_n (orthogonal the wall)

$q_n = q_x * n_x + q_y * ny$ (n_x, n_y components of the normal vector)

usually will jump. It is also, that the shear forces attain their maximum values near the walls. Therefore, the program calculates automatically the shear force in a distance 0.5 h (where h = width of the wall) on both sides of the wall.

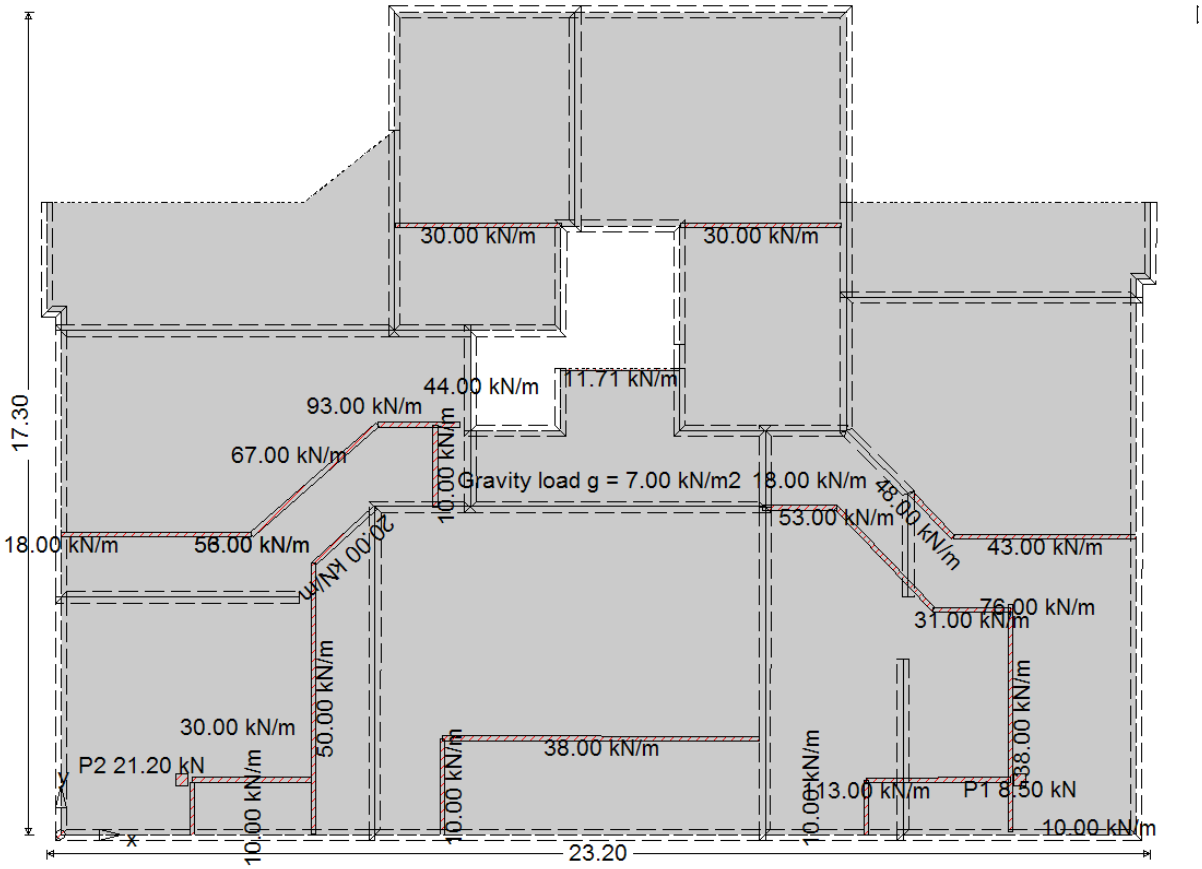
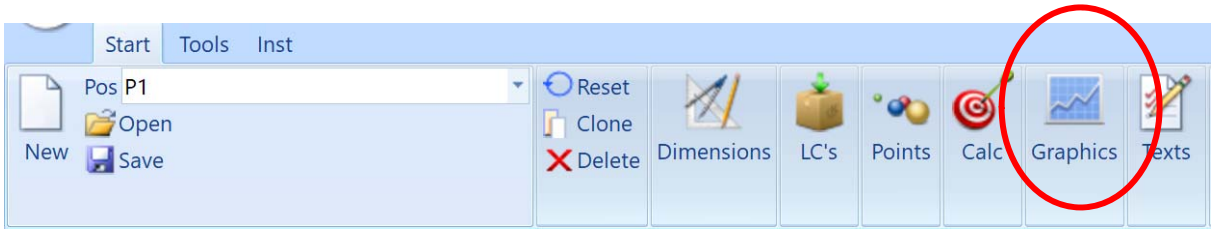


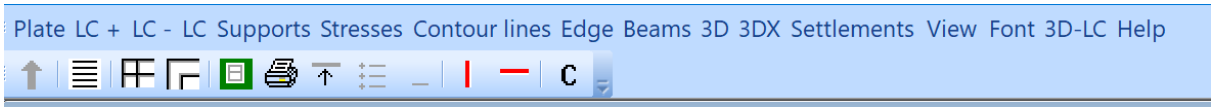
Fig. 8.5 Load

9. Display of results



To display the results, click on the button Graphics in the main menu.

This will open the program GRAPHICS program



where the

- **internal actions**
- **the principal moments**
- **the reinforcement and**
- **the deformations of the plate**

in form of

- **curves**
- **contour lines**
- **3-D representations**
- **diagrams**

9.1. Handling

Changing the scale

By pressing the plus and minus key on the number block, you can scale the drawing on the screen (principal moments, support reactions, etc.)

The same effects can be achieved with a wheel mouse by turning the wheel.

Changing the size of the font

The font size can be changed by pressing the shift-key and turning the mouse wheel.

Switching between load cases

With the arrow keys (up and down) you can switch between the load cases.

Zoom

To zoom in on a detail simply open a window with the mouse. The ESC-key takes you back to the previous view.

If you zoom in on the as-values you can pan the picture with the arrow keys on the keyboard.

Printing

Pictures can be printed, plotted or copied to the clipboard (icon C).

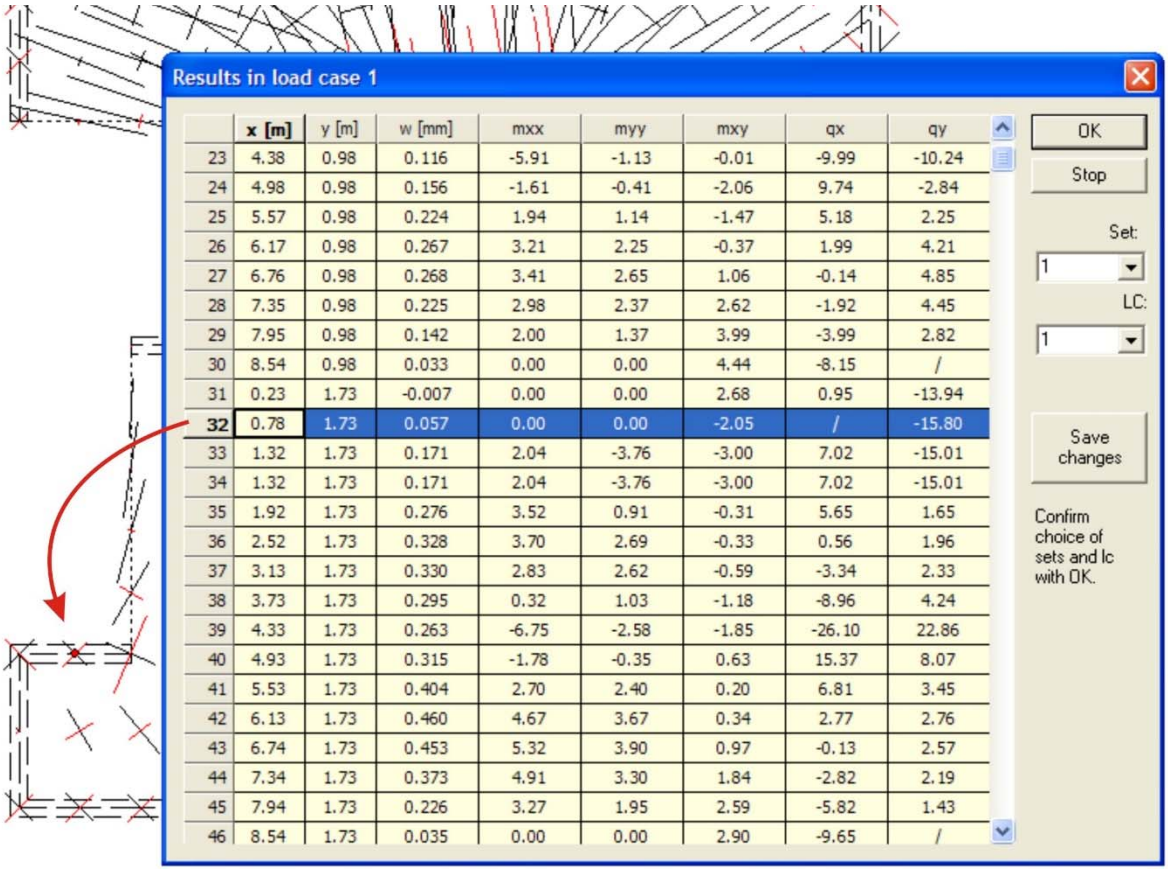
Click on the green edged printer icon for a print preview.

Context menus

In many windows, a click on the right mouse button will open a context menu

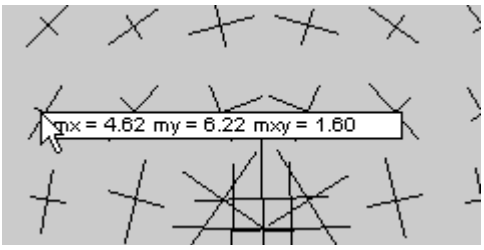
Tabular display

If you click on the leftmost column in a table (shaded grey in the following picture) in any row the position of the corresponding point with the coordinates x = …, y = …, will be displayed on the screen with a small red dot.



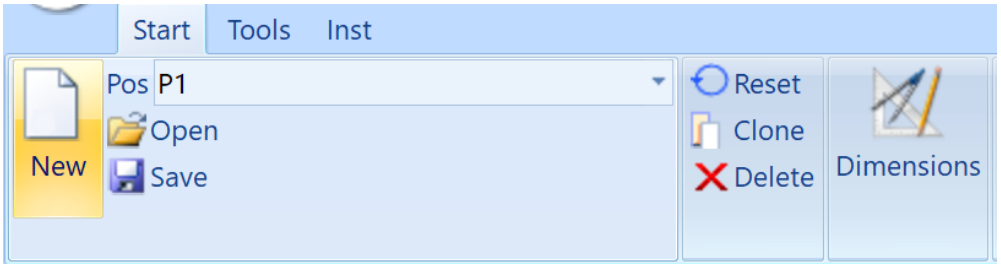
	x [m]	y [m]	w [mm]	mxx	myy	mxy	qx	qy
23	4.38	0.98	0.116	-5.91	-1.13	-0.01	-9.99	-10.24
24	4.98	0.98	0.156	-1.61	-0.41	-2.06	9.74	-2.84
25	5.57	0.98	0.224	1.94	1.14	-1.47	5.18	2.25
26	6.17	0.98	0.267	3.21	2.25	-0.37	1.99	4.21
27	6.76	0.98	0.268	3.41	2.65	1.06	-0.14	4.85
28	7.35	0.98	0.225	2.98	2.37	2.62	-1.92	4.45
29	7.95	0.98	0.142	2.00	1.37	3.99	-3.99	2.82
30	8.54	0.98	0.033	0.00	0.00	4.44	-8.15	/
31	0.23	1.73	-0.007	0.00	0.00	2.68	0.95	-13.94
32	0.78	1.73	0.057	0.00	0.00	-2.05	/	-15.80
33	1.32	1.73	0.171	2.04	-3.76	-3.00	7.02	-15.01
34	1.32	1.73	0.171	2.04	-3.76	-3.00	7.02	-15.01
35	1.92	1.73	0.276	3.52	0.91	-0.31	5.65	1.65
36	2.52	1.73	0.328	3.70	2.69	-0.33	0.56	1.96
37	3.13	1.73	0.330	2.83	2.62	-0.59	-3.34	2.33
38	3.73	1.73	0.295	0.32	1.03	-1.18	-8.96	4.24
39	4.33	1.73	0.263	-6.75	-2.58	-1.85	-26.10	22.86
40	4.93	1.73	0.315	-1.78	-0.35	0.63	15.37	8.07
41	5.53	1.73	0.404	2.70	2.40	0.20	6.81	3.45
42	6.13	1.73	0.460	4.67	3.67	0.34	2.77	2.76
43	6.74	1.73	0.453	5.32	3.90	0.97	-0.13	2.57
44	7.34	1.73	0.373	4.91	3.30	1.84	-2.82	2.19
45	7.94	1.73	0.226	3.27	1.95	2.59	-5.82	1.43
46	8.54	1.73	0.035	0.00	0.00	2.90	-9.65	/

If you click on the principle moments diagram the moments of the nearest stress points are displayed on the screen.



If you do the same when the reinforcement is displayed a table opens where the as-values are listed for this point.

10. STORING AND RETRIEVING



10.1. Storing a complete position

The program stores the files of a position, say position 180, in the subdirectory

SDIR180

which branches off from the actual path data as displayed in the main menu of the program.

To save the complete position you must manually store the whole subdirectory on an external drive or on a CD.

10.2. Storing only the input

Otherwise you can only store the input of a position. This will allow you to load the position from the archived files. But because in this procedure the binary files with the results of the previous analysis are not saved you have to redo the analysis (which should not take too long).

To store the input of, say position 180, click on the icon

and choose the drive or folder where you want to store the input.

The program stores the three echo files

```
ECHO_G.180.TXT        // geometry
ECHO_L.180.TXT        // load cases
ECHO_S.180.TXT        // stress points
```

in the folder. But not under their original names but under the names

```
ACADGEO.180
ACADLAST.180
ACADPKT.180
```

10.3. Retrieving a position

To retrieve a position, say 200, open the Windows-Explorer find the file

ACADGEO.200
and drag it onto the open program window.

This works also with the original echo-file

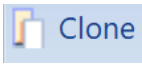
ECHO_G.200.TXT

In each case the program will not only load the file ACADGEO.200 but also the files ACADLAST.200 and ACADPKT.200 or ECHO_G.200.TX and ECHO_L.200.TXT and ECHO_S.200.TXT respectively.

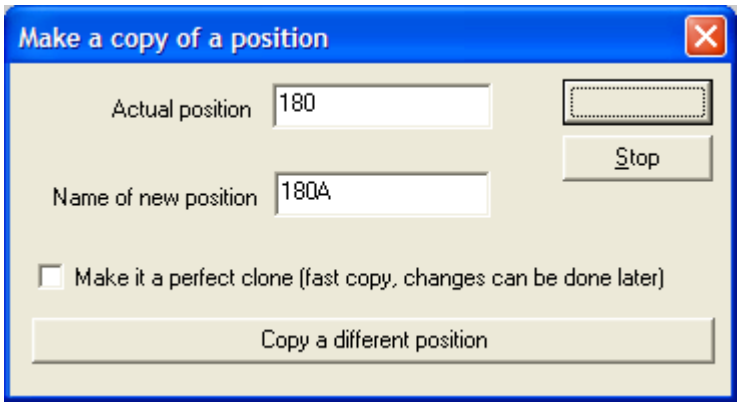
That is for a complete restore of a position all three archive files are necessary. Otherwise only the shape of the plate will be reconstructed from ACADGEO.200.

10.4. Duplicating a position

To duplicate a position, say 180, click on the icon



and enter a new name for the copy



The following happens:

1. The program generates a subdirectory SDIR180A.
2. It copies the echo-files of position 180 into the new folder SDIR180A and renames the echo-files to acad-files.

Next the program loads the files ACADGEO.180A and displays the plate on the screen allowing you to make modifications of the newly generated copy of the original plate.

The load cases too are either a direct copy from the original position or you can modify the load cases or enter new load cases. The same holds true for the stress points.

10.5. Resetting a position

A click on the icon

will reset the actual position. That is all previous results are deleted.