

# BE-PLATES

Manual

July 2017

Friedel Hartmann

[www.be-statik.de](http://www.be-statik.de)

[www.winfem.de](http://www.winfem.de)

[hartmann@be-statik.de](mailto:hartmann@be-statik.de)

<b>1. INSTALLATION .....</b>	<b>4</b>
1.1. STARTING THE PROGRAM.....	4
1.2. THE MAIN MENU .....	4
1.3. POSITIONS.....	4
1.4. PATH DATA .....	5
1.5. FIRST STEPS .....	7
<b>2. INTRODUCTION .....</b>	<b>1</b>
2.1. ENTERING THE SHAPE .....	1
2.2. LOADS .....	5
2.3. INPUT OF STRESS POINTS .....	8
2.4. ANALYSIS.....	10
2.5. RESULTS .....	10
<b>3. PLATE.....</b>	<b>1</b>
3.1. DIALOG INPUT MODE .....	1
3.2. GRAPHICAL INPUT MODE .....	4
3.3. DXF-FILES (AUTOCAD 2012).....	4
3.4. MATERIAL .....	6
3.5. SYSTEM OF COORDINATES .....	9
3.6. SIDES.....	9
3.7. BOUNDARY ELEMENTS .....	10
3.8. GENERATING THE BOUNDARY ELEMENTS .....	10
3.9. OPENINGS.....	10
3.10. CIRCULAR ARCS.....	11
3.11. POINT SUPPORTS .....	11
3.12. TRANSVERSE SUPPORTS .....	12
3.13. CONTINUOUS BEAMS.....	14
<b>4. LOAD .....</b>	<b>16</b>
4.1. THE SIGN .....	16
4.2. GRAVITY LOAD.....	17
4.3. LINE LOADS .....	18
4.4. POINT LOADS .....	18
4.5. EDGE LOADS .....	19
4.6. PARTIAL AREA LOADS.....	20
4.7. SETTLEMENTS OF POINT SUPPORTS .....	21
4.8. DISLOCATIONS.....	22
<b>5. STRESS POINTS .....</b>	<b>1</b>
5.1. NEW SET .....	1
5.2. GRID.....	1
5.3. SECTIONS.....	3
5.4. LINES.....	3
5.5. POINTS.....	4
<b>6. RESULTS.....</b>	<b>1</b>
6.1. EQUILIBRIUM.....	1
6.2. INTERNAL ACTIONS.....	3
6.3. BOUNDARY VALUES .....	3
<b>7. DISPLAY OF RESULTS.....</b>	<b>5</b>
7.1. HANDLING.....	5
7.1.1. <i>Boundary elements</i> .....	8
7.1.2. <i>Boundary values</i> .....	8
7.1.3. <i>Stresses and displacements</i> .....	9

- 8. STORING AND RETRIEVING..... 1**
- 8.1. STORING A COMPLETE PROJECT..... 1
- 8.2. STORING ONLY THE INPUT..... 1
- 8.3. RETRIEVING A PROJECT..... 2
- 8.4. DUPLICATING A PROJECT..... 2
- 8.5. RESETTING A PROJECT..... 3

## 1. INSTALLATION

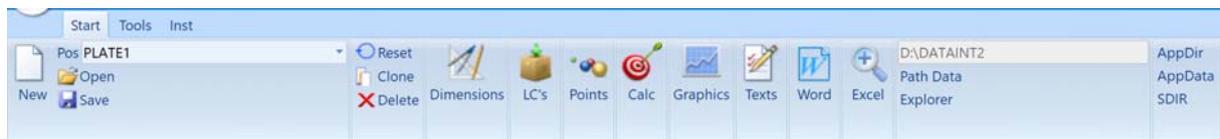
### 1.1. Starting the program

To start the program click on the file

BE-PLATES.EXE

in the program directory or on the screen.

### 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 at which the program outputs the displacements and stresses.

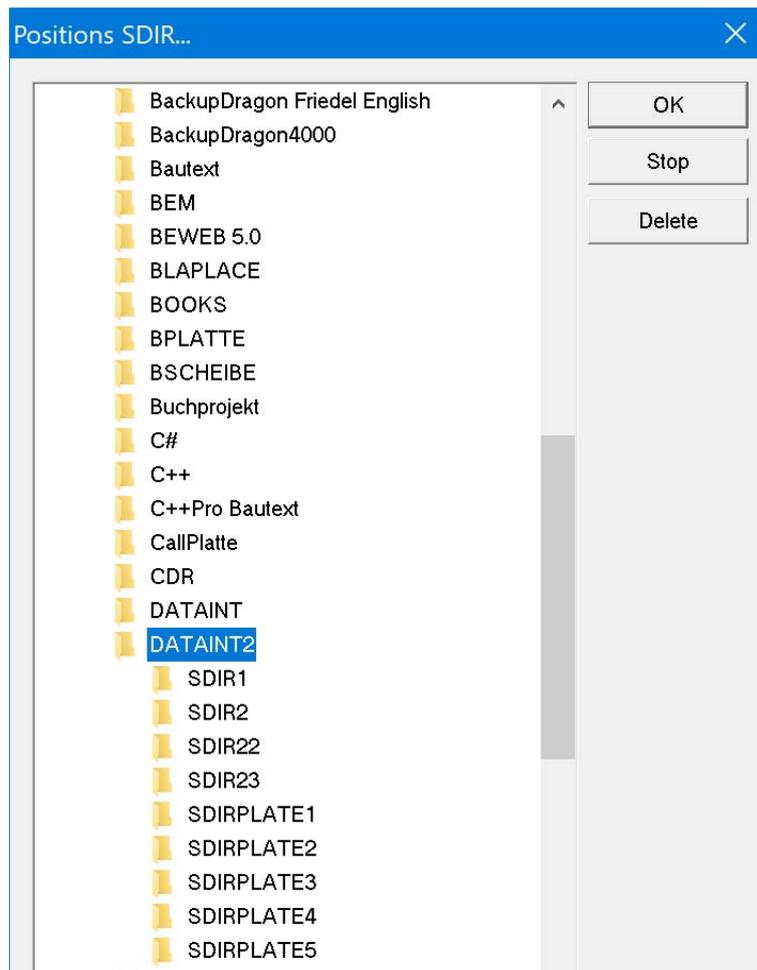
### 1.3. Positions

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.



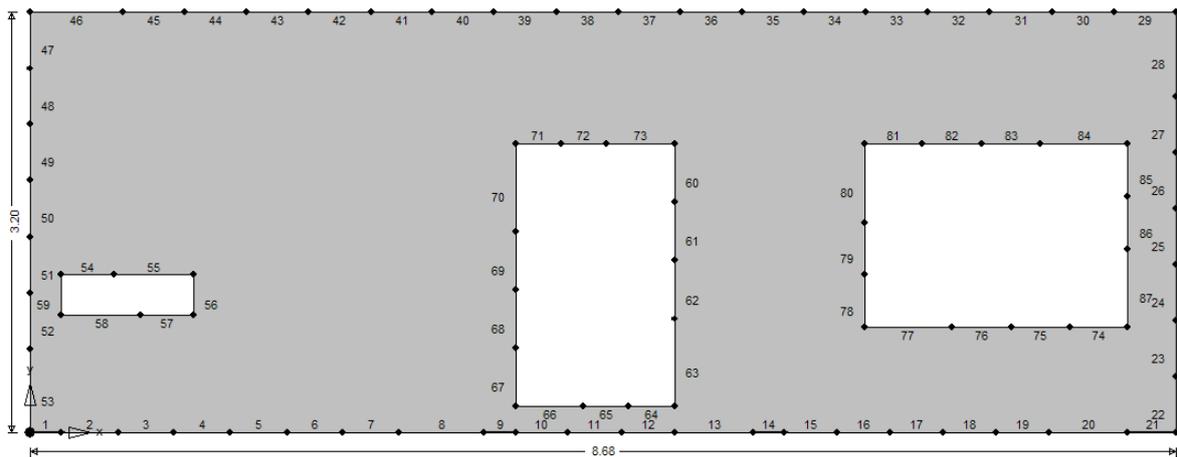
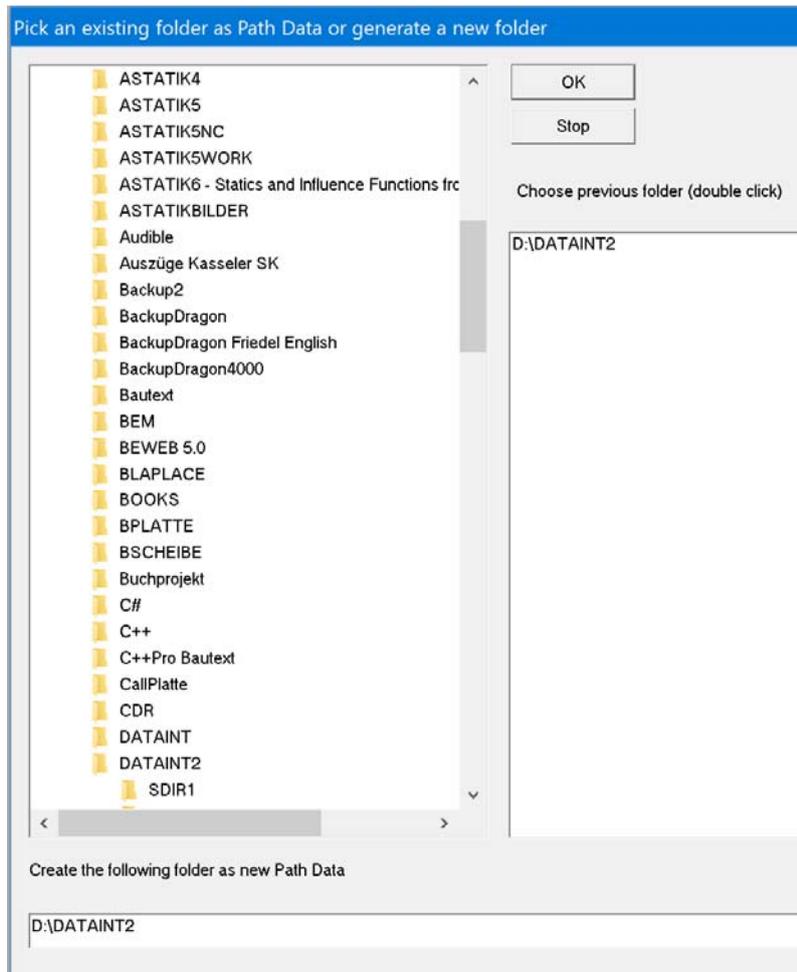
#### 1.4. Path Data

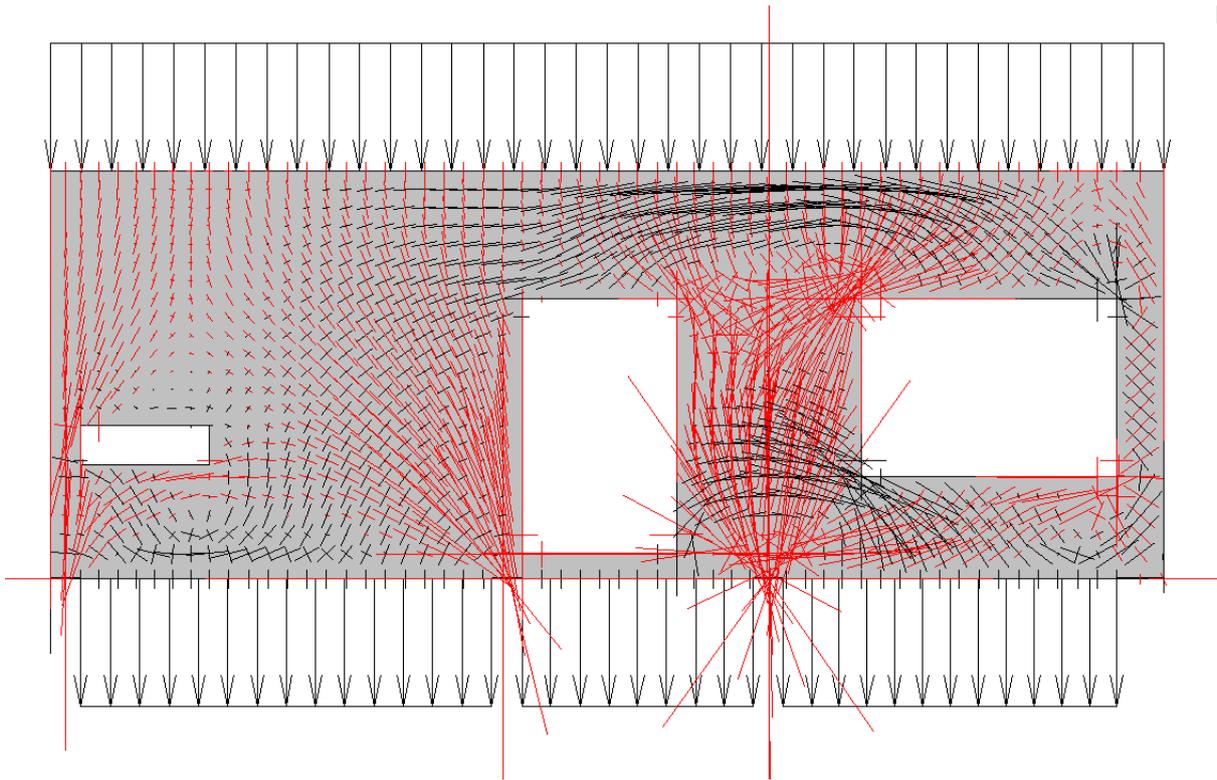
The data which belong to position 100 are stored in the folder SDIR100 and the data which belong to position 200 are stored in the folder SDIR200. The root directory for these folders is what *path data* points to. It can be any directory on the hard drive or on a network.

To change the path, click on the button

**Path Data**

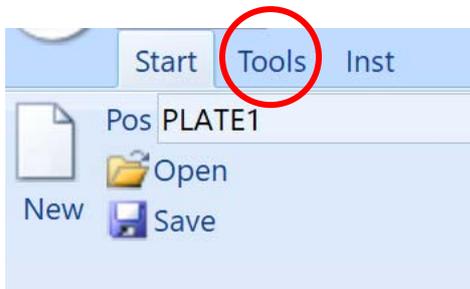
in the main menu.





### 1.5. First steps

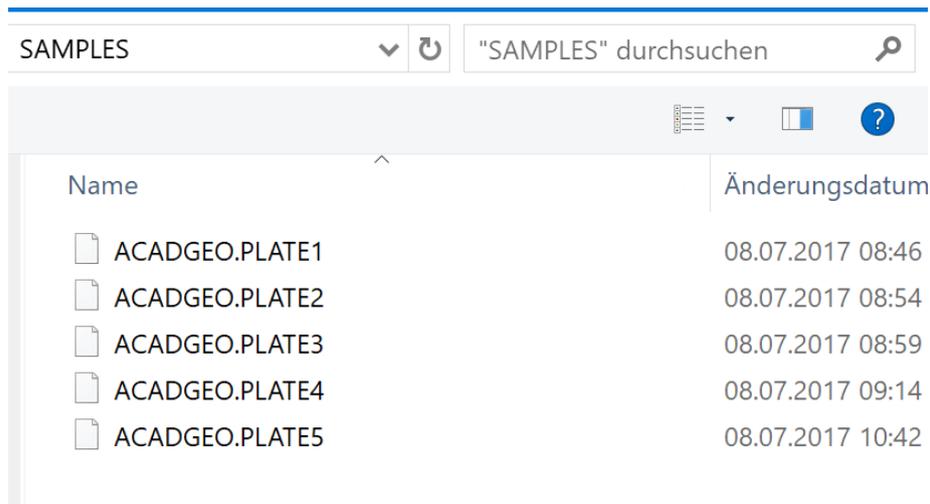
1. Click on the button Tools



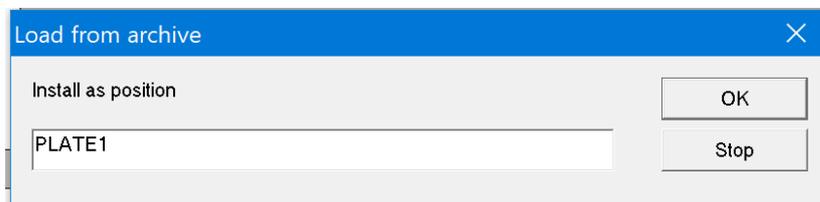
- and click on the icon Load samples



This should open the folder SAMPLES in the folder where the program is stored



3. Double click on the file ACADGEO.PLATE1 and confirm your choice

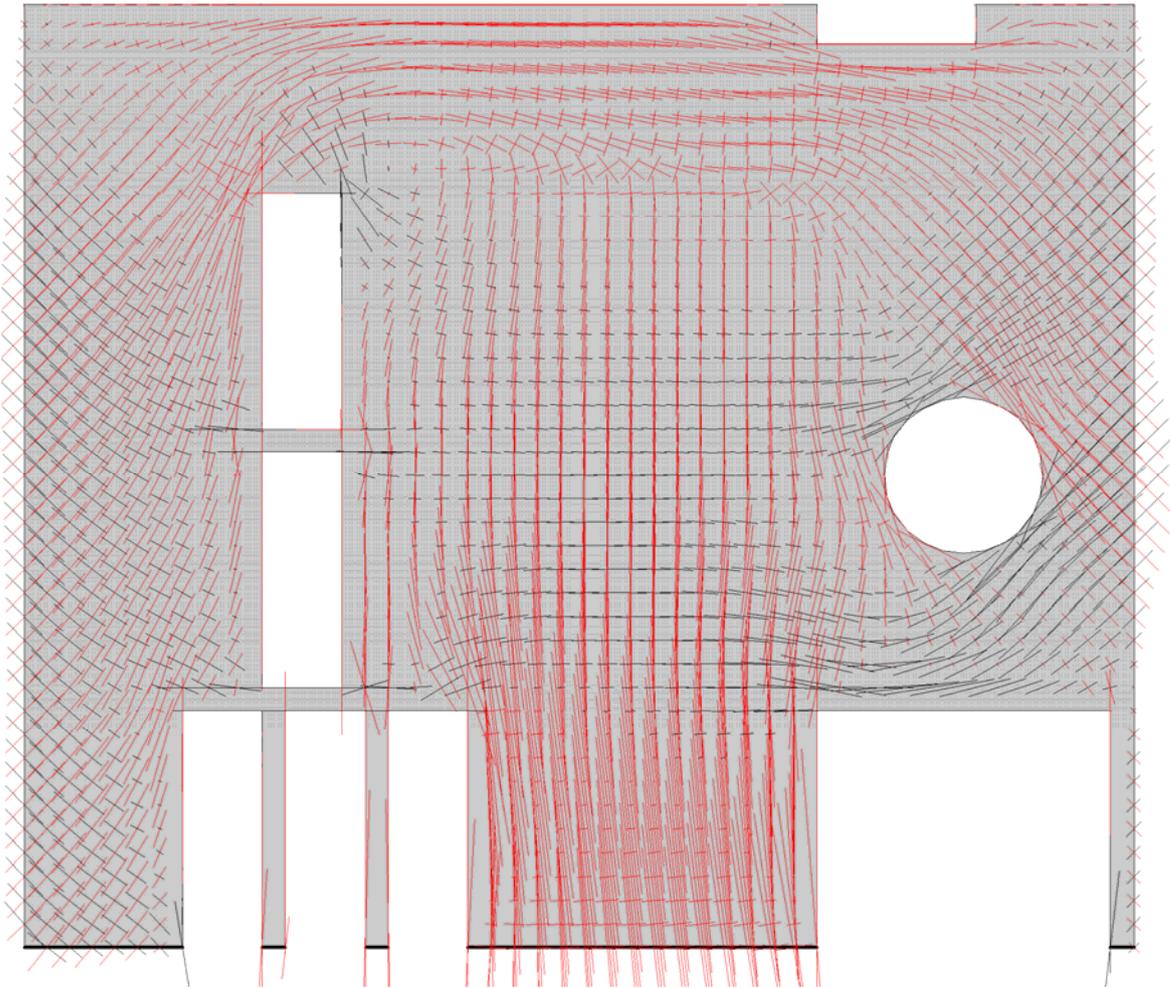


The program then reads the input from these three files and automatically discretizes the plate, inputs the load cases and loads the stress points.

You only have to click on the Calc icon to start the analysis of the plate



and when the analysis is done then click on the Graphics 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

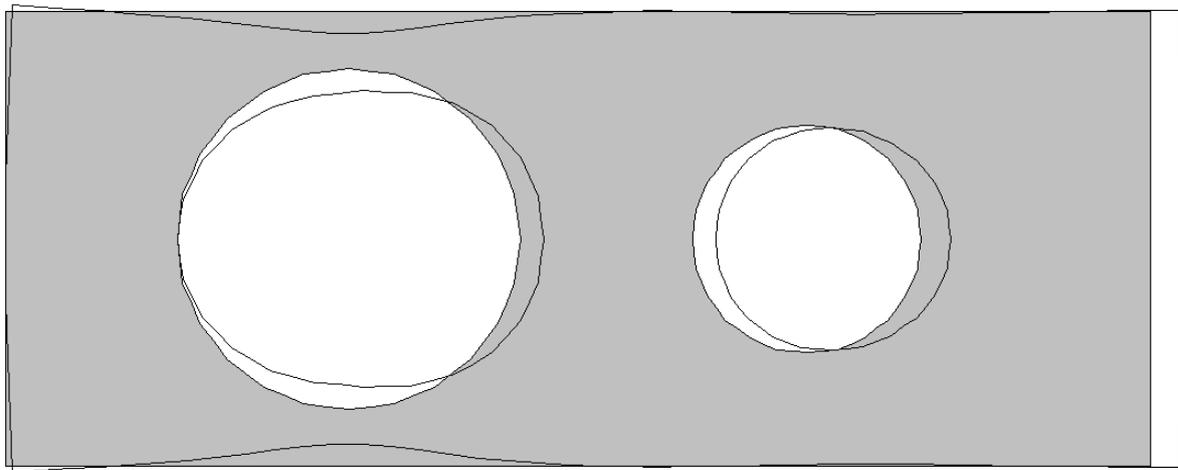
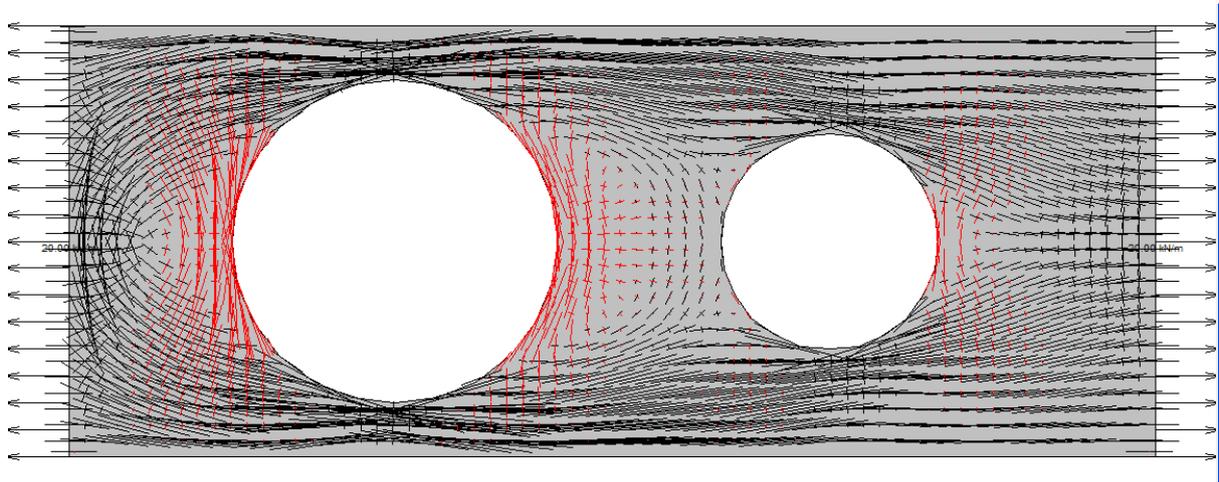
Ctrl + Mouse wheel = Font size

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.

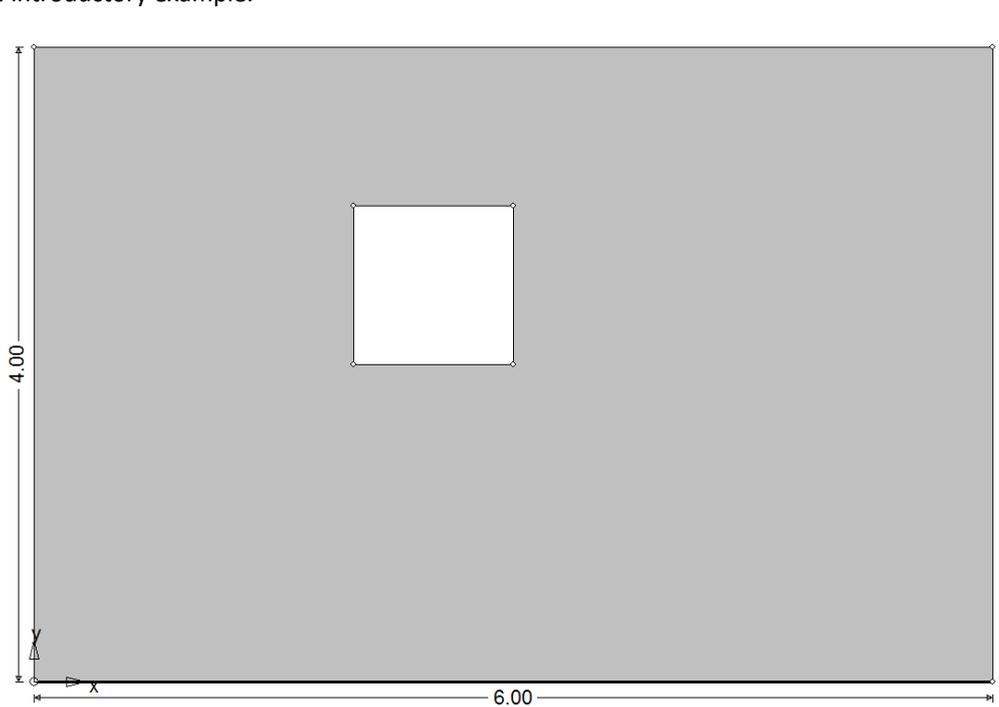


## 2. INTRODUCTION

A simple free standing wall

size : 6 x 4 m  
thickness: 30 cm  
concrete: C 20 ( $E = 3.0 \cdot 10^7 \text{ kN/m}^2$ )  
Poisson's ratio = 0.16  
fixed along the lower edge  
gravity load + line loads + ...

serves as introductory example.



### 2.1. Entering the shape

We first click on the button New



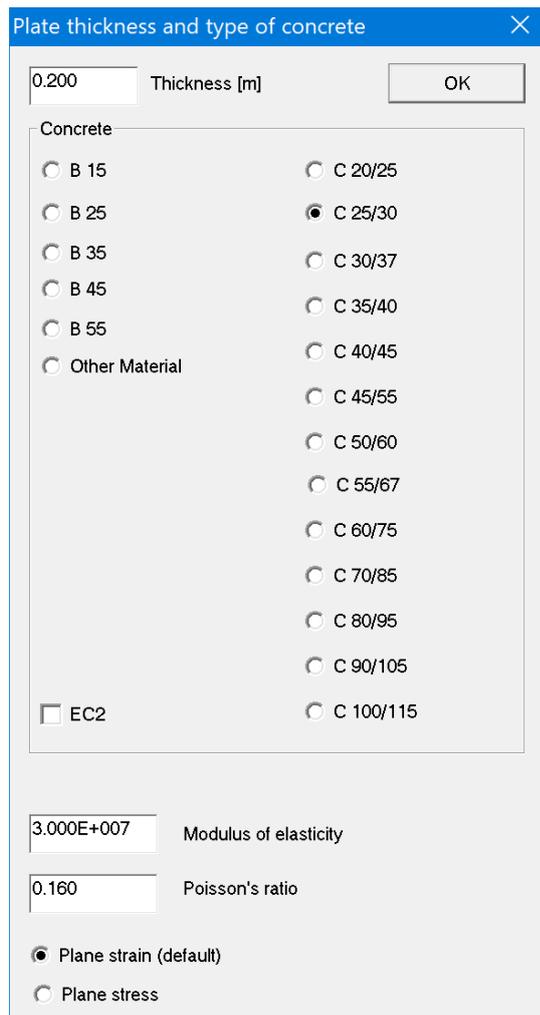
and enter the name of the position

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

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

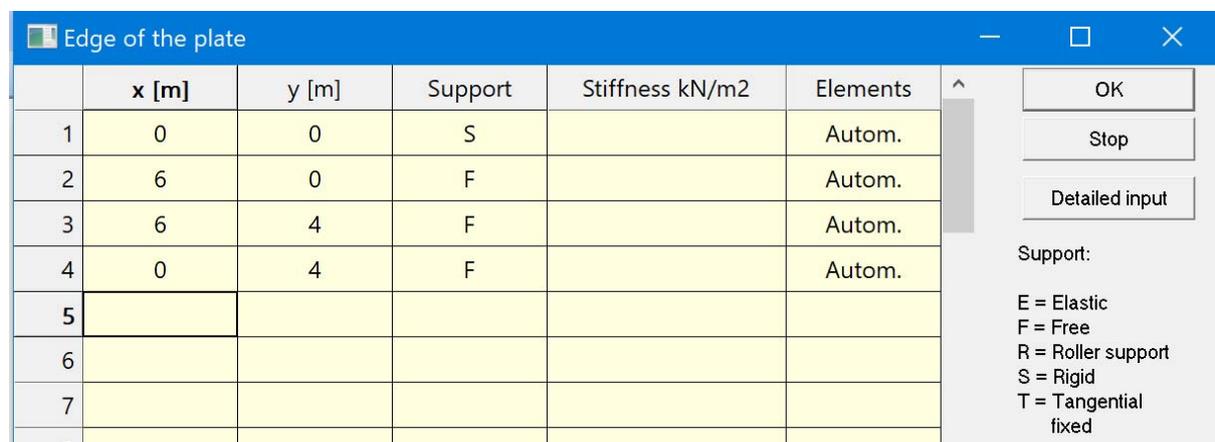
First we enter the title of the new position

set the element length to 0.5 m and specify the plate thickness



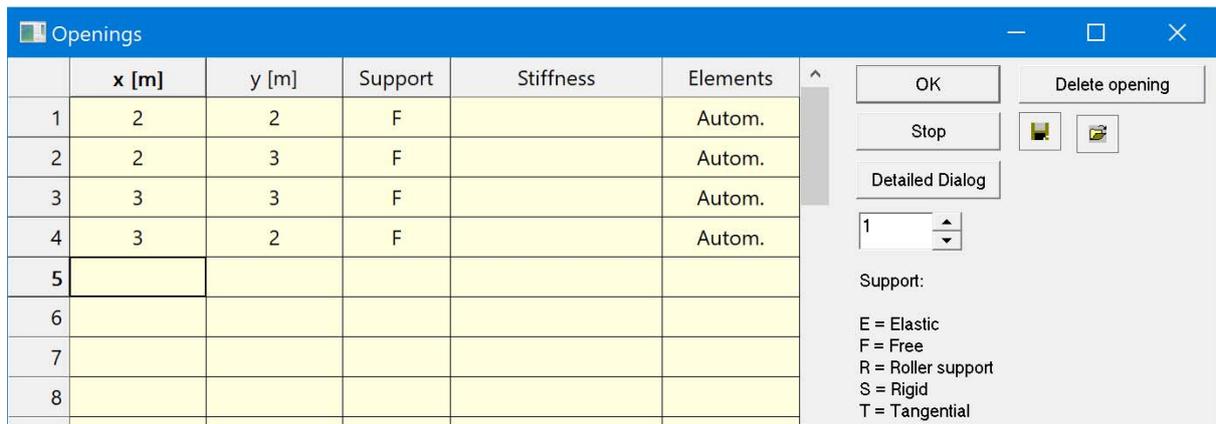
Enter the thickness of the plate and choose a grade C 20/25 concrete: The numeric value of the modulus of elasticity is predefined. To sidestep the preselection choose 'Other material' and enter your choice of the modulus E.

Close the dialog and next click on the button **Edge** to enter the coordinates of the four corners of the plate.



The lower side of the plate is clamped (S = fixed) while the other sides are not restrained (F = free).

Next click on the button **Openings** to enter the four corners of the opening.



Note that the sense of rotation is opposite to the sense of rotation on the outer edge of the plate.

Next call the discretization of the plate by clicking on the icon



which you find in the menu list on top of the main window.

After a short break you'll see a log file with an echo of the input and a 3-D model of the plate is displayed.

```

D:\DATAINT2\SDIRD1\D1.TXT
Plate analysis with BE-PLATES Version 12.3
-----
Wall plate
-----
Date: Sunday 9. 7. 2017 D:\DATAINT2\SDIRD1
-----
Concrete C 25/30 E = 26700.0 MN/m2 (DIN 1045-1)
Poisson's ratio = 0.16
Thickness = 0.20 m
Plane stress
E' = 26192.0 MN/m2 nue' = 0.14
Element length = 0.50 m

Edge
----
      Side      Elements  Node  Supp.
      x [m]  y [m]  qty. Length  Number  cond.
      [kN/m2]  Form
-----
1  0.00  0.00  12  0.50  1  25  rigid  S
2  6.00  0.00  8  0.50  25  41  free  S
3  6.00  4.00  12  0.50  41  65  free  S
4  0.00  4.00  8  0.50  65  1  free  S

Size = 23.00 m2

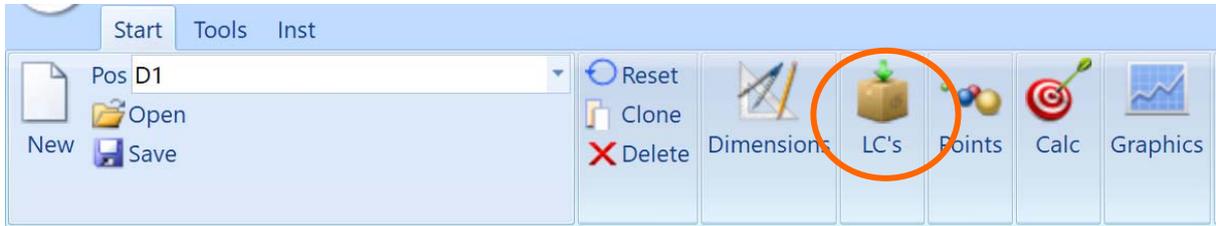
Opening 1
-----
      Side      Elements  Node  Supp.
      x [m]  y [m]  qty. Length  Number  cond.
      [kN/m2]  Form
-----
1  2.00  2.00  2  0.50  81  85  free  S
2  2.00  3.00  2  0.50  85  89  free  S
3  3.00  3.00  2  0.50  89  93  free  S

```

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

## 2.2. Loads

In the main menu click on the entry **LC's** to specify the loading.



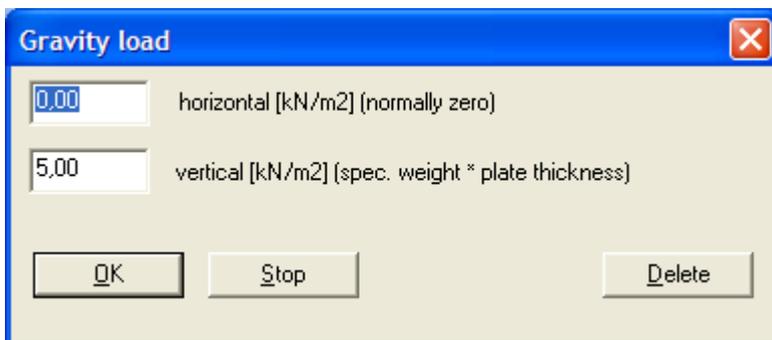
The following menu will be displayed.



We start with the dead weight of the plate. To enter this load case click on the entry

Gravity load

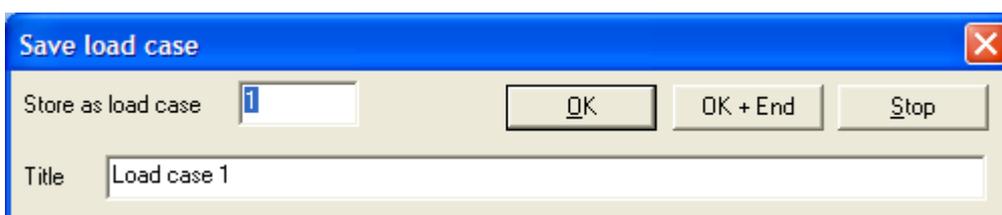
and confirm the preselection



Next store 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.

In load case 2 a vertical edge load acts at the upper edge of the plate and also the opening.

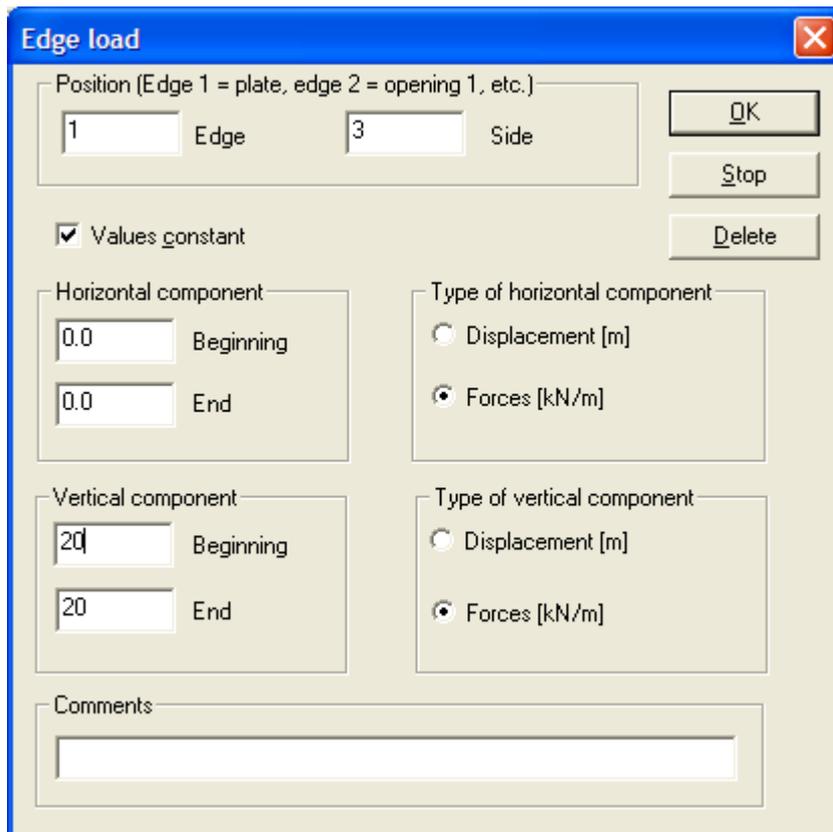
To reset the input and to clear the screen click on the icon New



To start the input of the edge load click on the icon



and next on the upper edge of the plate. In the dialog

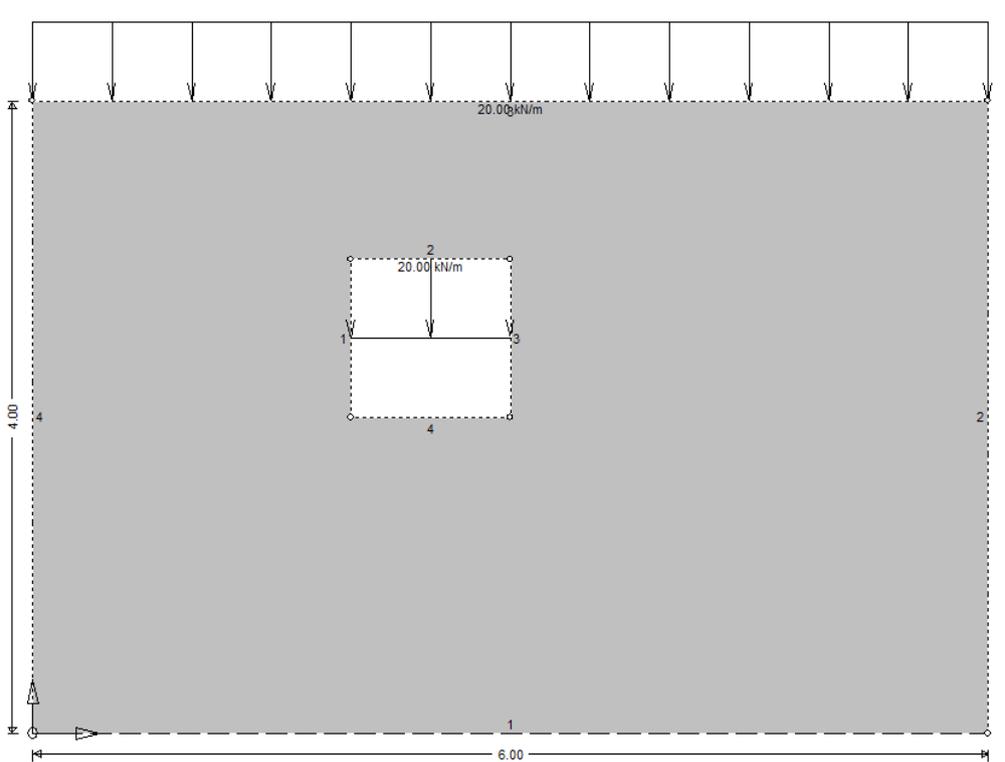


The image shows a dialog box titled "Edge load" with a blue title bar and a close button (X) in the top right corner. The dialog is divided into several sections:

- Position (Edge 1 = plate, edge 2 = opening 1, etc.):** Two input fields. The first is labeled "Edge" and contains the value "1". The second is labeled "Side" and contains the value "3".
- Buttons:** Three buttons are located on the right side: "OK", "Stop", and "Delete".
- Values constant:** A checked checkbox labeled "Values constant".
- Horizontal component:** Two input fields. The first is labeled "Beginning" and contains "0.0". The second is labeled "End" and contains "0.0".
- Type of horizontal component:** Two radio buttons. "Displacement [m]" is unselected, and "Forces [kN/m]" is selected.
- Vertical component:** Two input fields. The first is labeled "Beginning" and contains "20". The second is labeled "End" and contains "20".
- Type of vertical component:** Two radio buttons. "Displacement [m]" is unselected, and "Forces [kN/m]" is selected.
- Comments:** A text area at the bottom for entering additional information.

enter a value of 20 kN/m as vertical edge load. Click on OK and the dialog will close.

Next click with the mouse on the upper edge of the opening. The same edge load as before will appear. To edit the edge load click the right mouse button and a context will allow you to specify additional details.



Store the load case by clicking on the icon

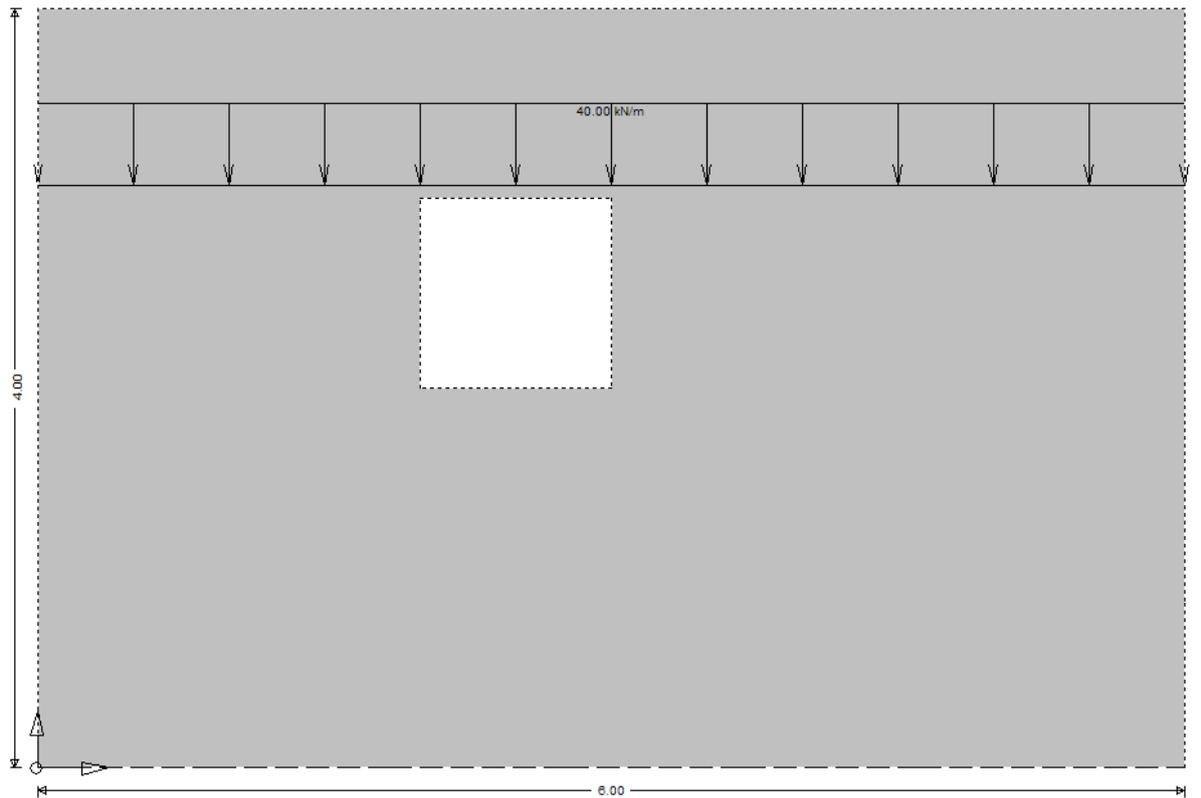


and then clear the plate by clicking on the icon new



to prepare for the input of the next load case.

In the third load case a vertical line load is acting across the plate.



To prepare for the input click on the entry **Line load** in the menu and specify the coordinates of the end points and the magnitude of the load

Line loads									
	xa [m]	ya [m]	xb [m]	yb [m]	Value px_a	Value px_b	Value py_a	Value py_b	Comments
1	0	3.5	6	3.5	0	0	40	40	
2									
3									
4									

Store the load case as load case # 3. This ends the input of the loading. 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 stresses and the displacements. Because there is no mesh these points must be generated separately in a BE-program. Note that the accuracy does not depend on the mesh width that is the mesh (or grid of points) can have any shape, any size and any distance between the points

Stress points come in different sets.

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

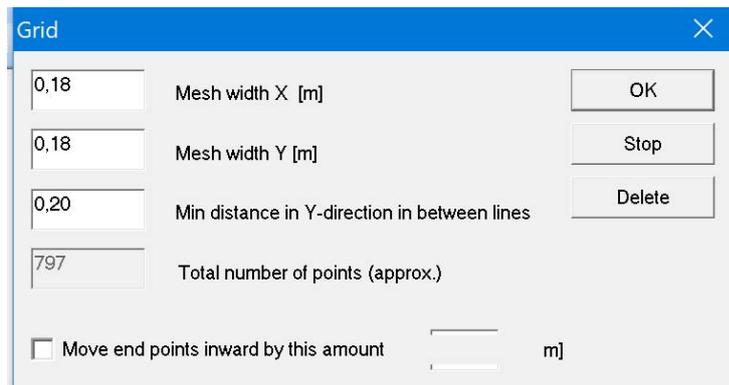
#2: Set number two may consist of a set of single lines of stress points which allow a closer inspection of certain regions of a plate.

#3: Set number three may consist of single stress points where additional information about the stress field is needed.

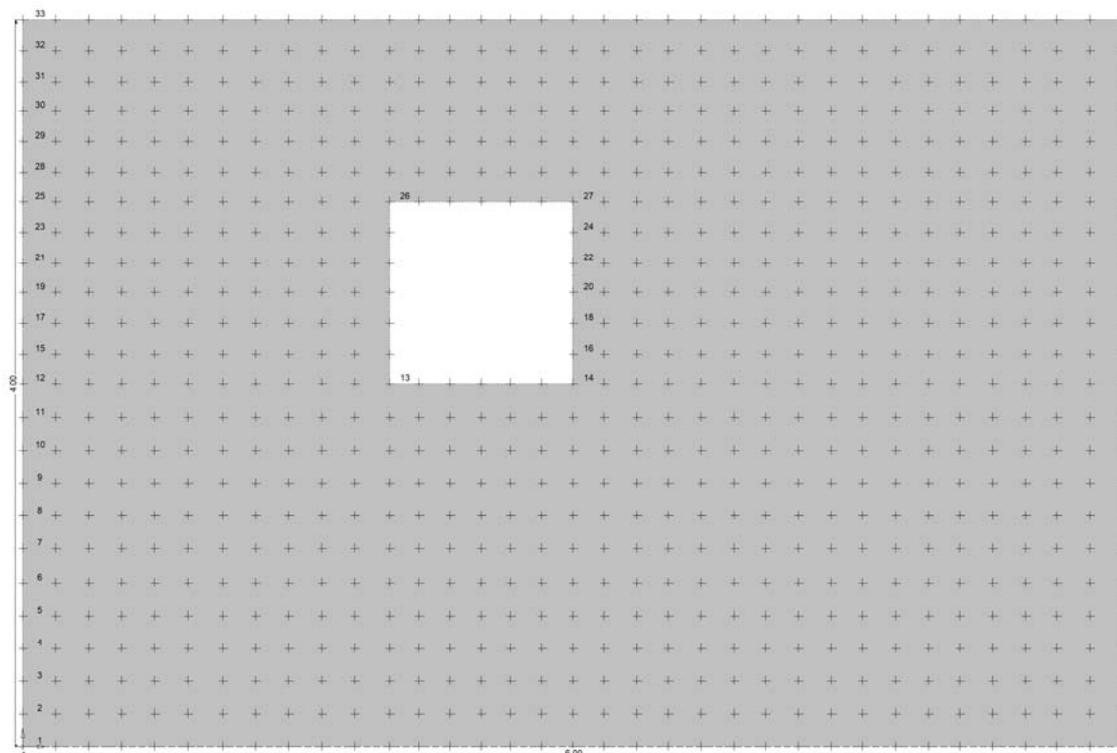
To start the input of the stress points click on the entry **P**oints in the main menu and in the menu



click on the icon  to cover the plate with a grid of stress points. Choose a mesh width of 0.18 m!

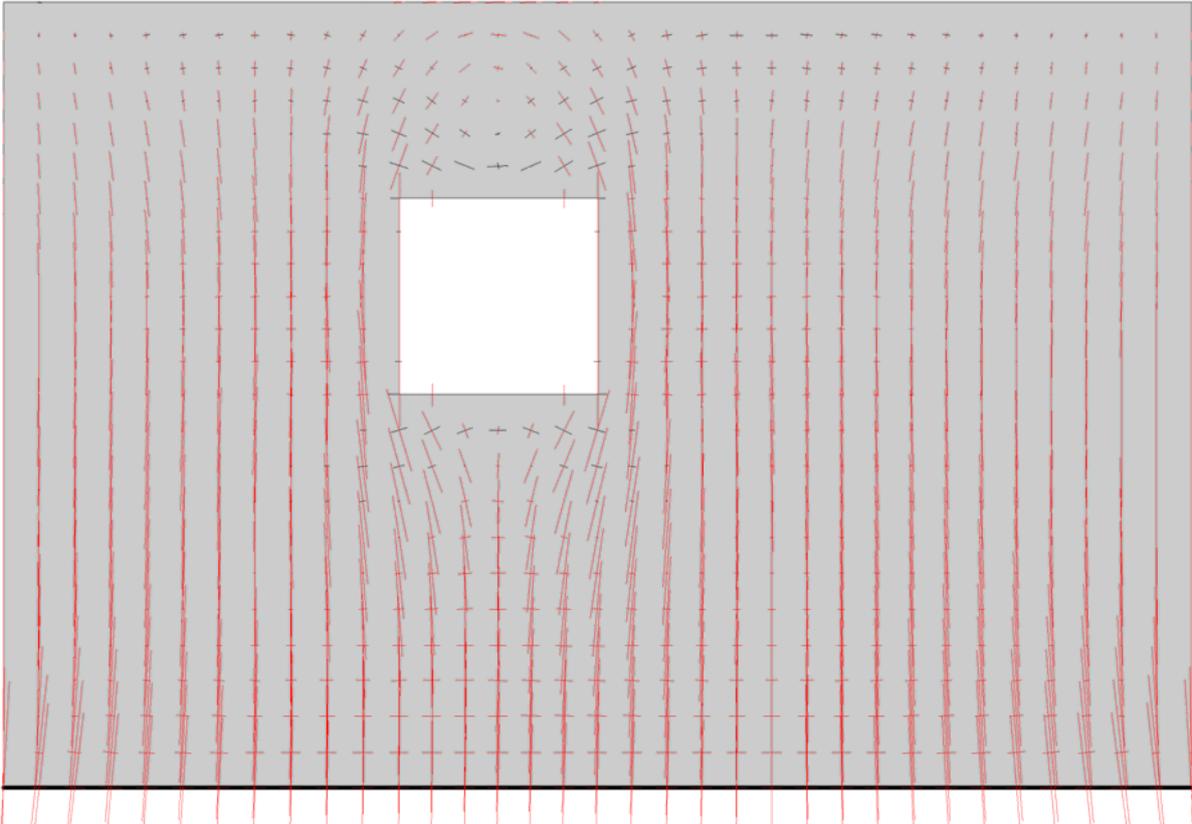


and let the program generate the grid.

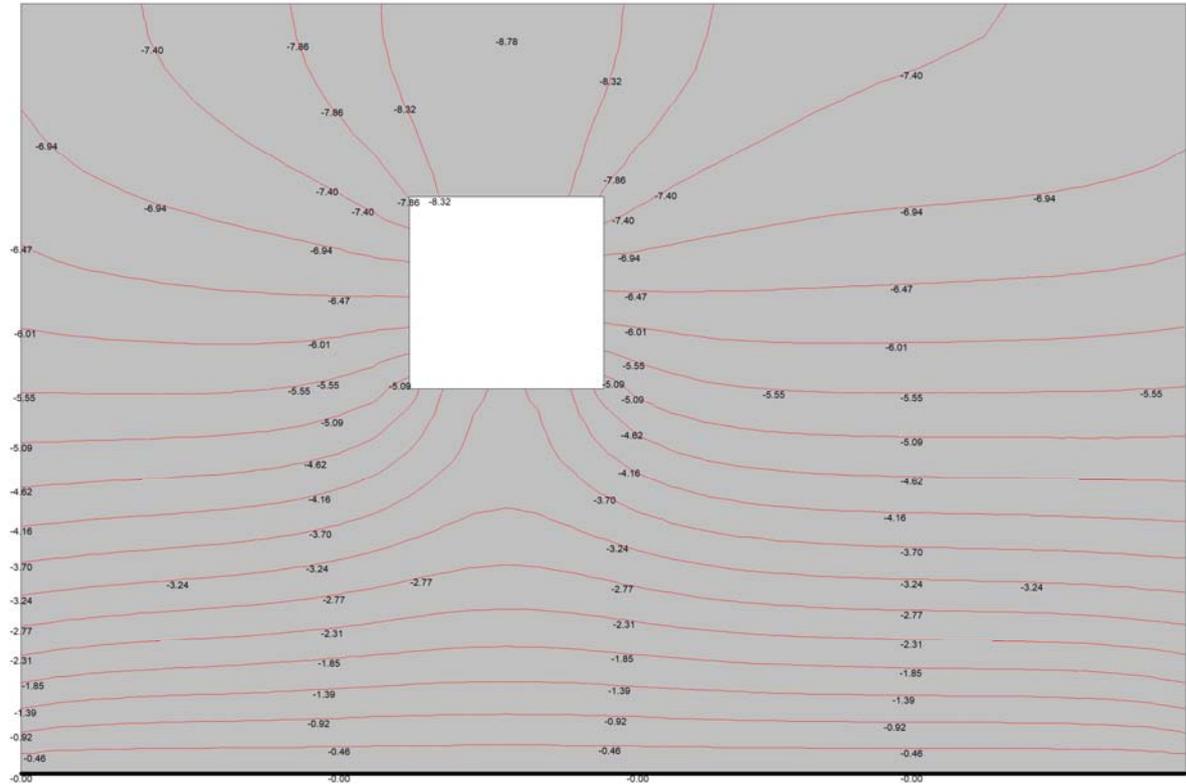


Store the grid by clicking on the icon 





or for example the contour lines of the vertical displacement





### 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 mode click on the entry Dimensions in the program menu.

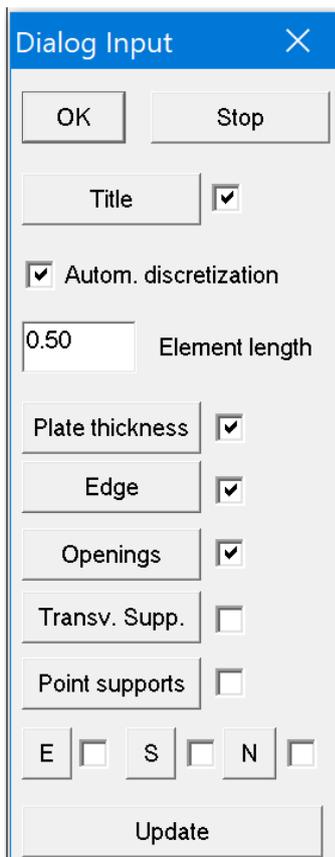
#### 3.1. Dialog input mode

Dialog input mode is simple and easy to use. It is recommended for novice users. To start this input mode click on the corresponding entry in the menu bar.



In this mode you use standard dialogs and tables to detail the shape of the plate. The usual procedure is that 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 Update 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.

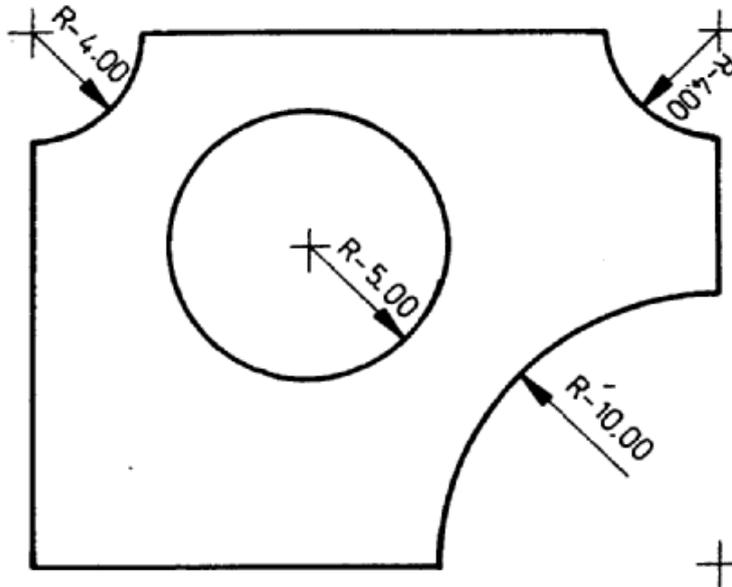


Fig. 3.1 Circular arcs can be entered by specifying the coordinates of the first point of the arc and the radius  $R$  of the arc. (Edge -> Detailed input)

Edge of the plate							
	x [m]	y [m]	Form	Radius	Support	Stiffness	Elements
1	0.00	0.00	S		F		Autom.
2	15	0.00	C	-10	F		
3	25	10	S		F		
4	25	16	C	-4	F		
5	21	20	S		F		
6	4	20	C	-4	F		
7	0	16	S		C		
8							
9							
10							
11							
12							
13							
14							
15							
16							
17							
18							
19							
20							

Support:

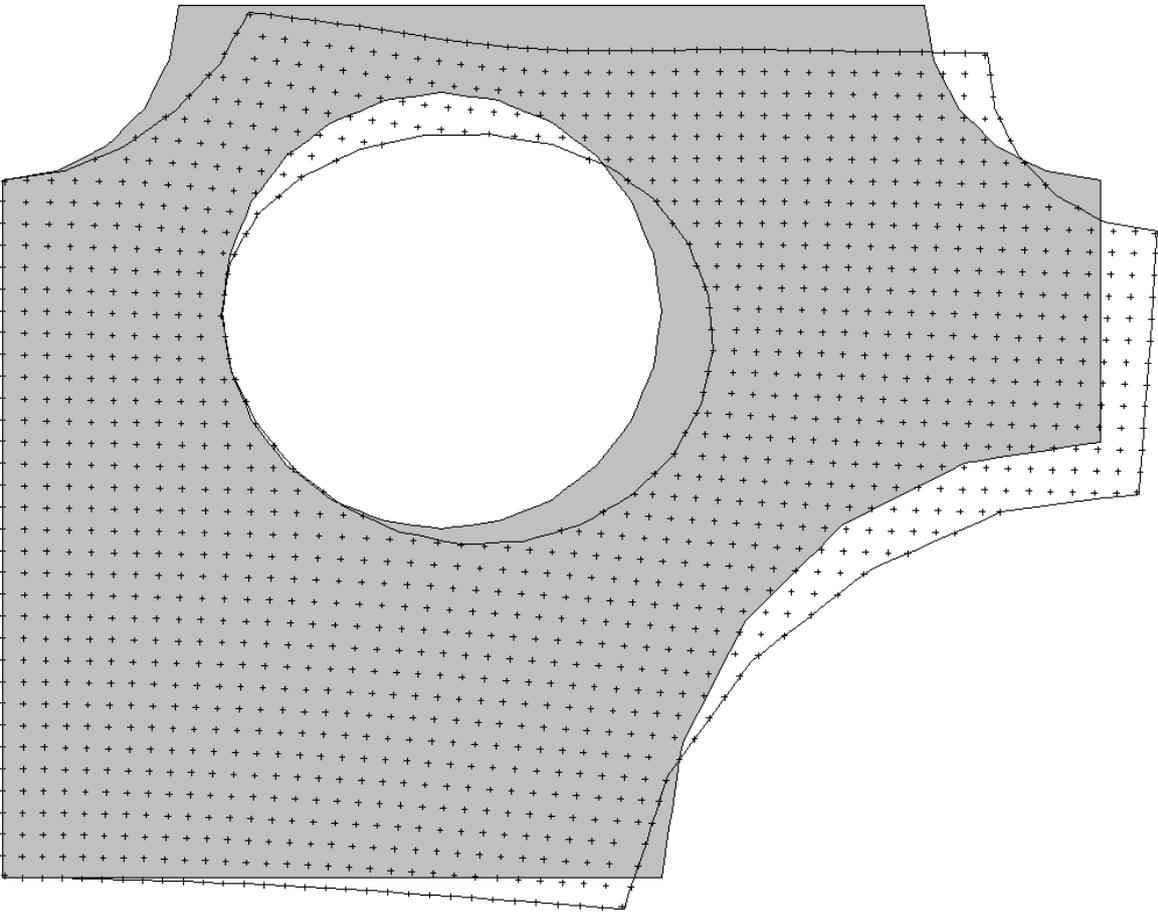
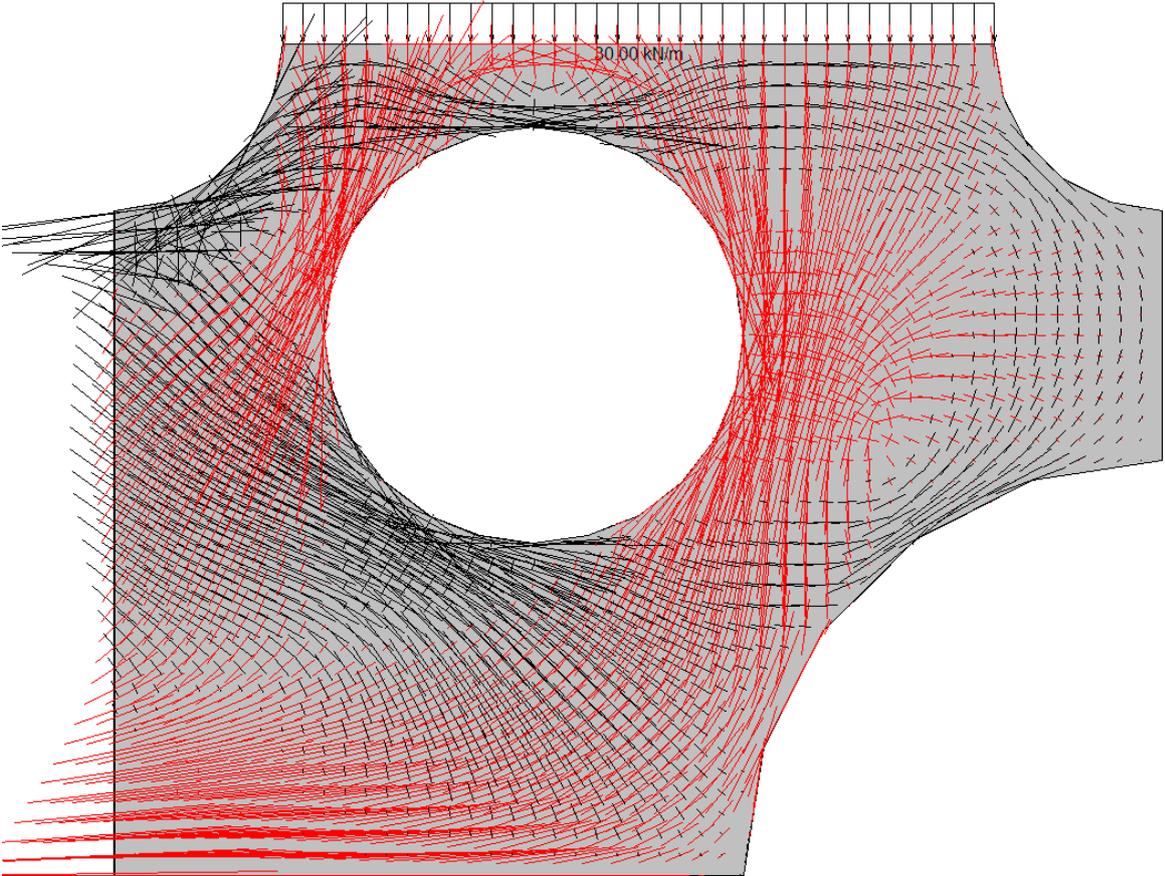
C = Clamped  
 E = Elastic  
 F = Free  
 R = Roller  
 T = Tangential support

Ctrl + C = Copy  
 Ctrl + V = Paste  
 Ctrl + X = Delete  
 Ctrl + I = Insert row  
 Ctrl + L = Delete row

Form:

C = curved  
 S = straight

Print



### 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.

To start click on the icon of the object you want to draw

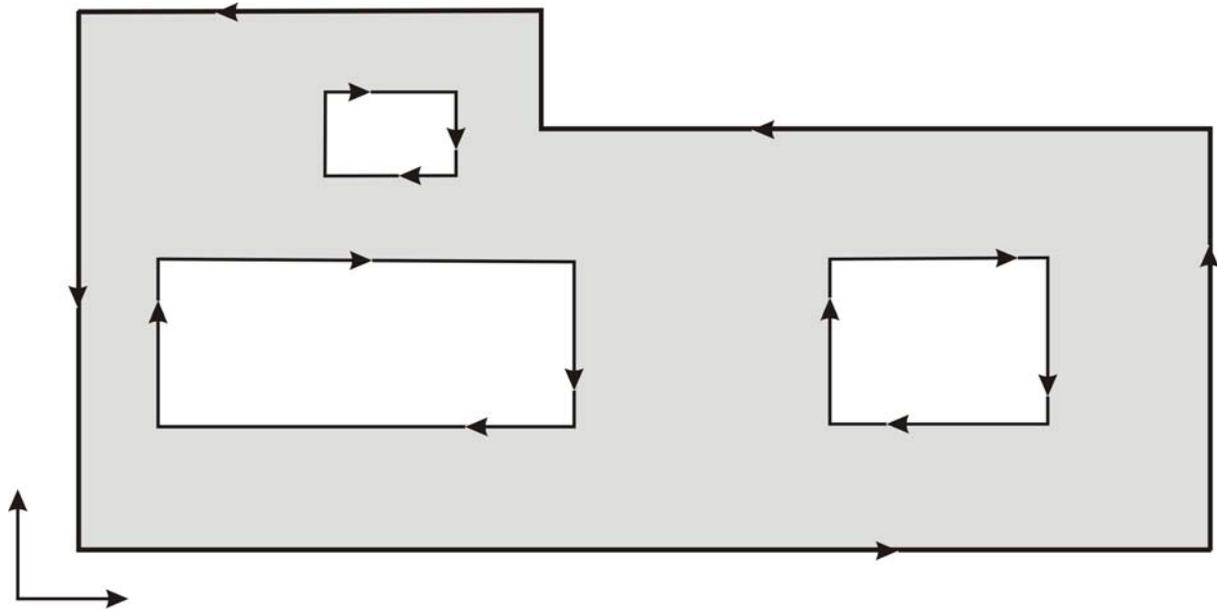


Fig 3.2 The sense of rotation along the edge of an opening is opposite to the outer edge..

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

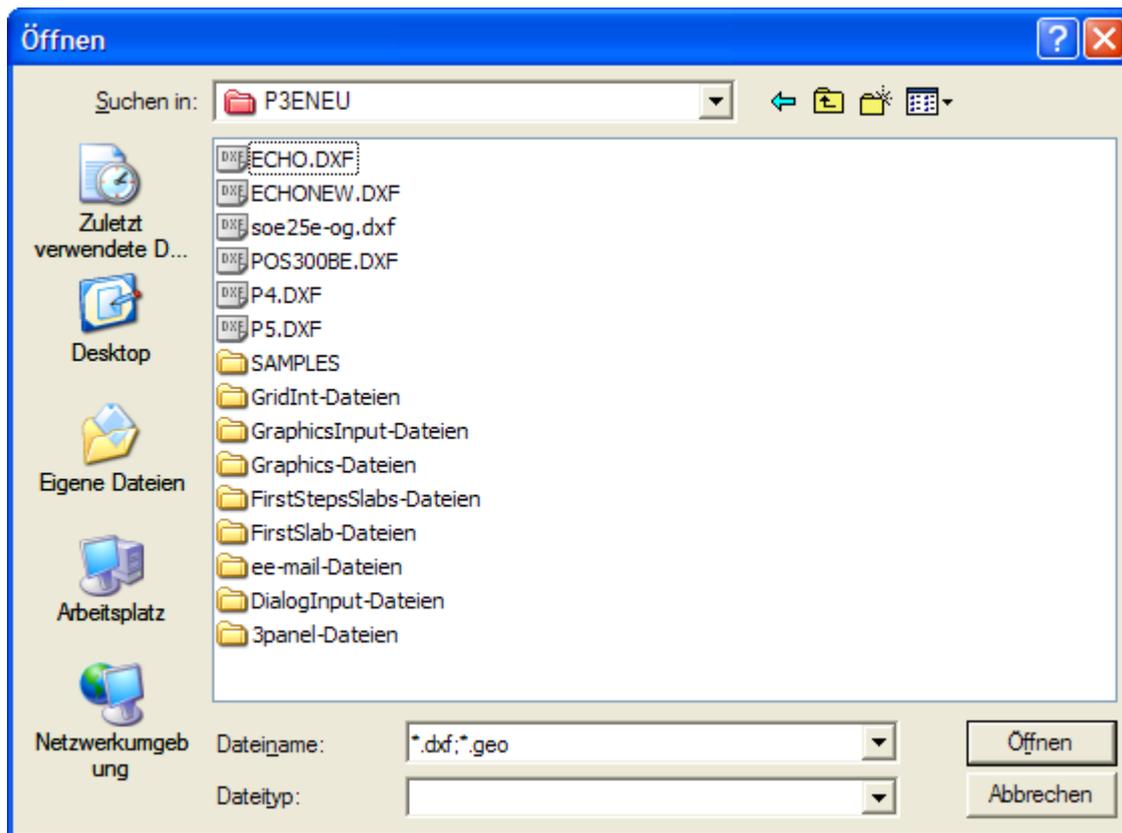


The subdivision of the edges and walls into boundary elements, etc. is done by the program.

### 3.3. dxf-files (Autocad 2012)

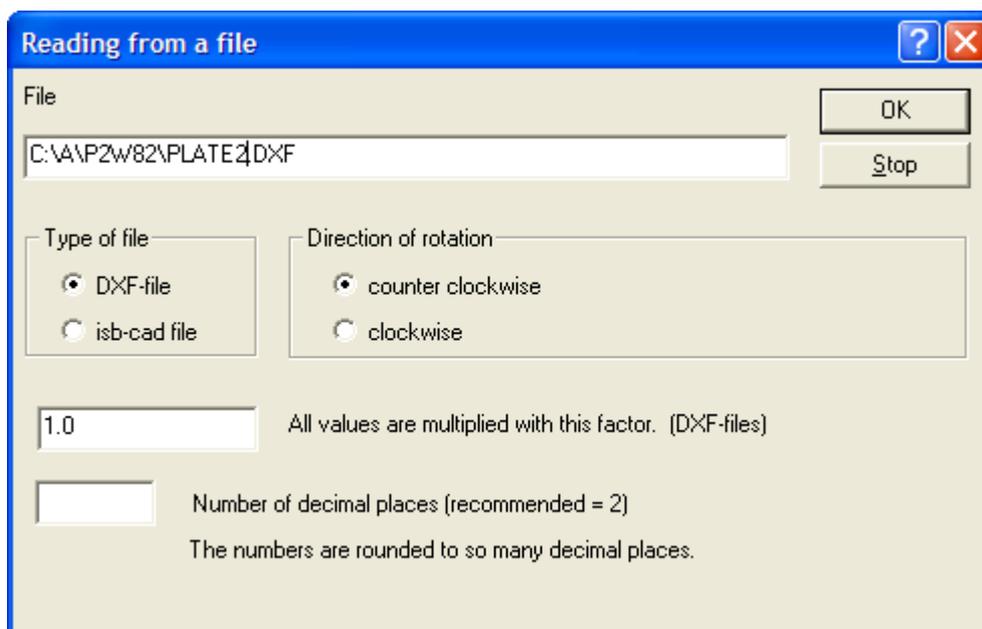
If the coordinates of the plate are stored in a dxf-file you can retrieve the coordinates from these files.

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 actually has filtered out from the input stream.

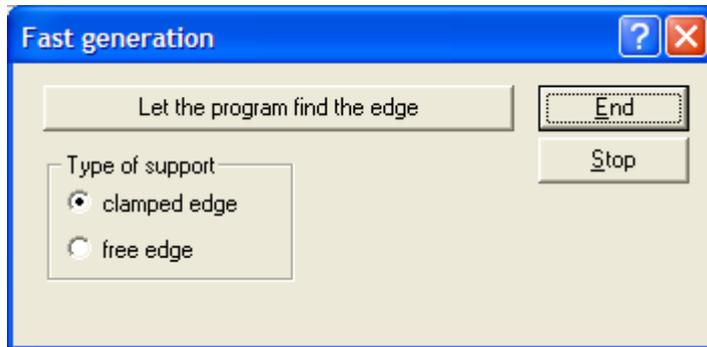
The program will generate from the input stream the edge of the plate. To succeed it is important that the program knows the sense of rotation of the plate that is in which sequence the coordinates were stored in the dxf-file.



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. Next specify the type of support along the edge of the plate. This can be changed later manually.

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

### 3.4. Material

#### Units

The program works with the units Meter and Kilo Newton

Unit of length = m  
Unit of forces = kN

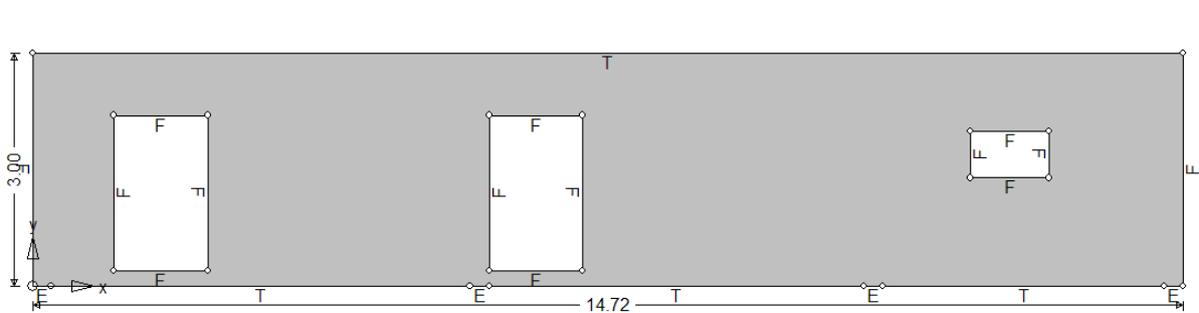


Fig. 3.8 F = Free edge, T = tangential support (in the direction of the edge), E = elastic support

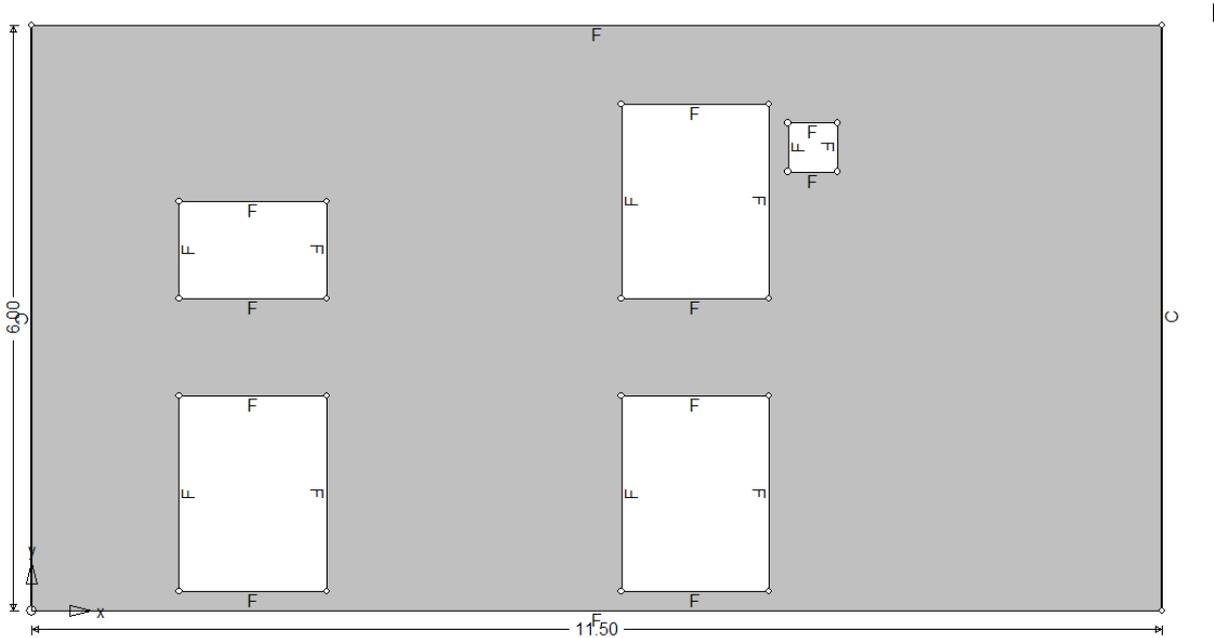


Fig. 3.9 Shear wall hanging between two walls on the left and on the right (not shown). The boundary condition is specified as S = fixed (clamped)

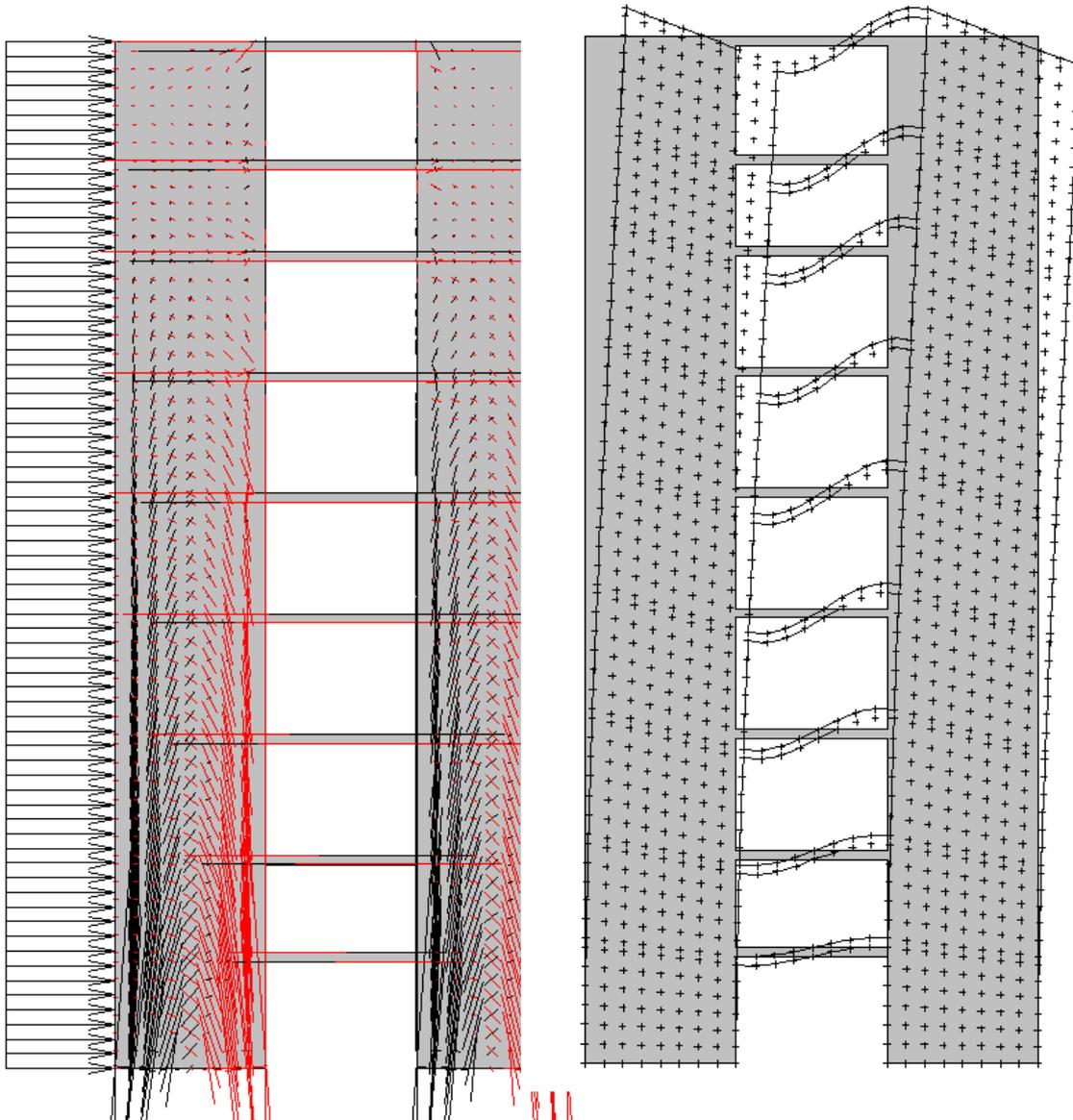


Fig. 3.10 Effects of a lateral load on two coupled twin towers

### Modulus of elasticity

According to the German building code DIN 1045-1 the modulus of elasticity of a grade CX/Y concrete is

C20/25	$E = 2.49 \cdot 10^7 \text{ kN/m}^2$
C25/30	$E = 2.67$
C30/37	$E = 2.83$
...	
C 0	$E = \text{defined by the user}$

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/m <sup>2</sup> ]	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.

## 3.5. System of coordinates

The y-axis points upward and the x-axis to the right.

The sense of rotation is counter-clockwise for the edge and counter-clockwise for openings.

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

## 3.6. 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

<b>Fixed</b>	the displacements are zero
<b>free</b>	free edge
<b>elastic</b>	elastic support of the edge (normal to the edge)
<b>roller support</b>	no restraints parallel to the edge
<b>tangential</b>	fixed parallel to the edge

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.

Supports
?
×

Plate edge

<input checked="" type="radio"/>	Rigid	S	<input type="button" value="All sides"/>  <input type="button" value="End"/>
<input type="radio"/>	Free	F	
<input type="radio"/>	Elastic	E	
<input type="radio"/>	Roller support	R	
<input type="radio"/>	Tangential	T	

Stiffness [kN/m]

Support on the backside of the plate

Transverse bulkhead

Support stiffness

Support stiffness

### 3.7. Boundary elements

The edge of the plate is subdivided into quadratic boundary elements. The subdivision is usually done by the program automatically, based on the average element length specified by the user, 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 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 wall 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.

### 3.8. 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.

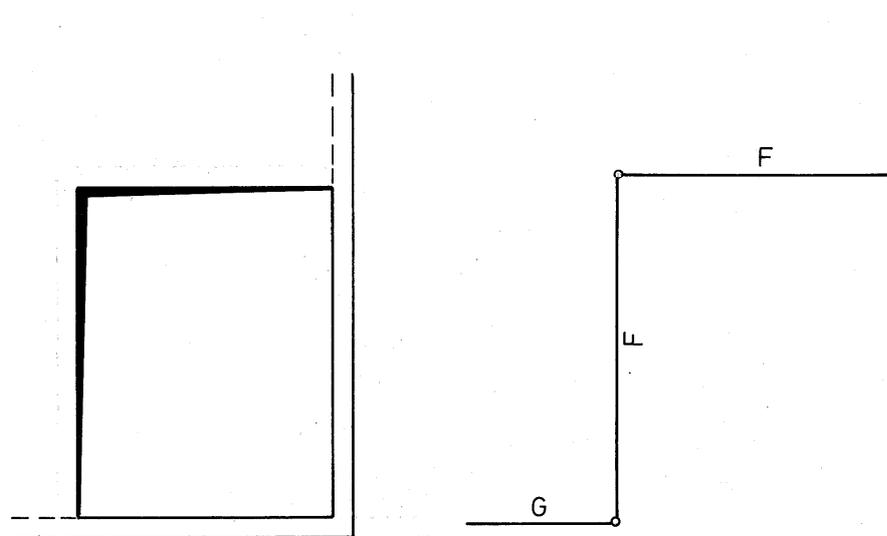


Fig. 3.17 Openings that border directly the edge of the plate are eliminated by moving the edge of the plate inward.

### 3.9. 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.

## 3.10. 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.

## 3.11. Point supports

These are single points which are restrained in x- or y-direction that is which work against elastic springs. In graphical input mode you click on the icon



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

**Point support**

1

2.3 x [m]

4.5 y [m]

Horizontal displacement fixed

Vertical displacement fixed

1.2E6 cx [kN/m] Horiz. stiffness

2.0E4 cy [kN/m] Vertic. stiffness

Store

Delete

End

you can specify the exact coordinates of the point and the stiffness of the point support. So a point that is fixed actually is working against elastic springs. Only if you choose large values for the stiffness of the support will the displacements of the 'fixed' point be zero.

Point supports should be located inside the plate. If necessary move point supports at the edge of the plate inward by 0.1 m. The reason is that the support reactions are modeled as 'full space' point forces and not as 'half space' point forces (Boussinesq).

### 3.12. Transverse Supports

Some plates, for example curtain walls, are not supported alongside their edge but rather cling to line supports on their backside. These line supports are what we call transverse supports

They are specified by the coordinates of the end points and the stiffness in horizontal ( $c_x$ ) and vertical ( $c_y$ ) direction.

**Transvers supports (on the backside of the plate)**

	xa [m]	ya [m]	xb [m]	yb [m]	Stiffness kx	Stiffness ky	Elements
1	1	1	4	1	123456	123456	Autom.
2							
3							
4							
5							

OK

Stop

Print

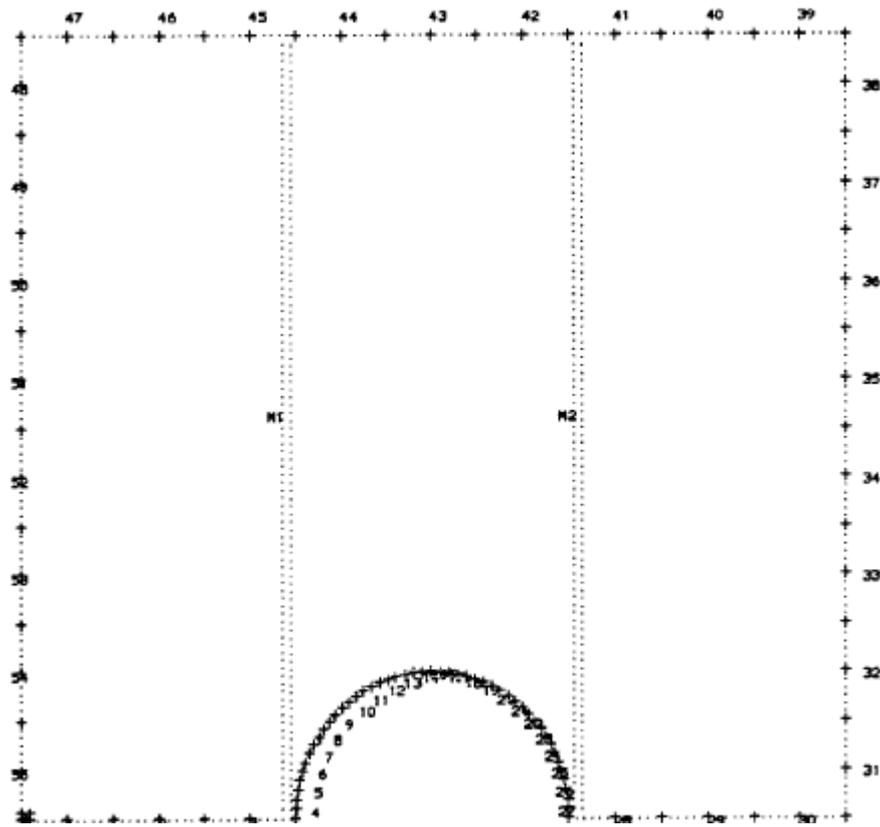


Fig. 3.7 Subdivision of the edge into boundary elements

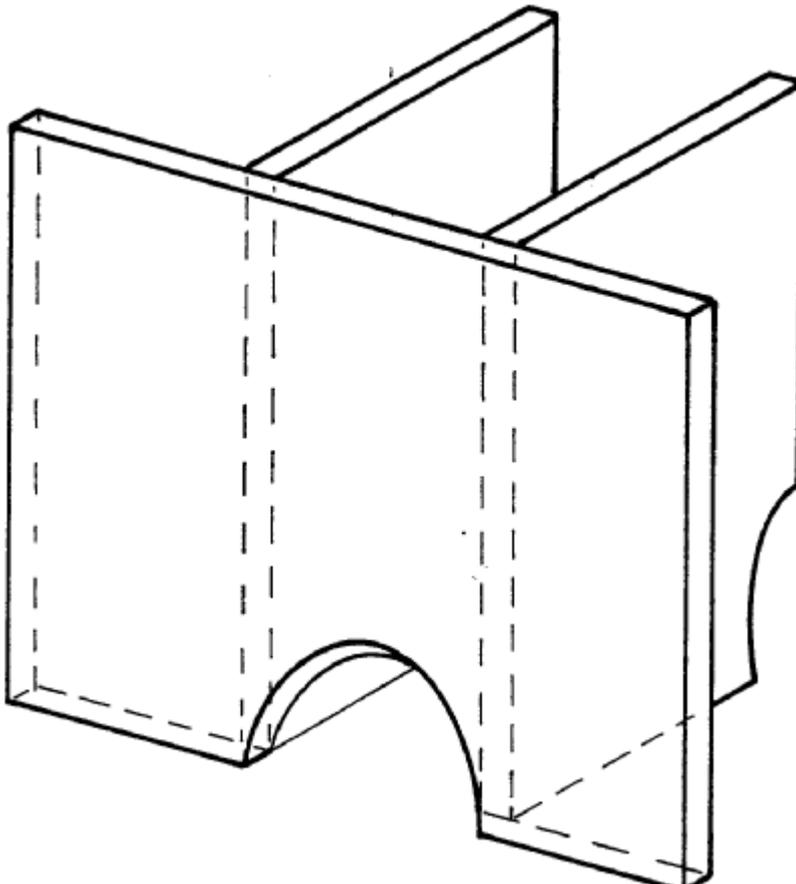


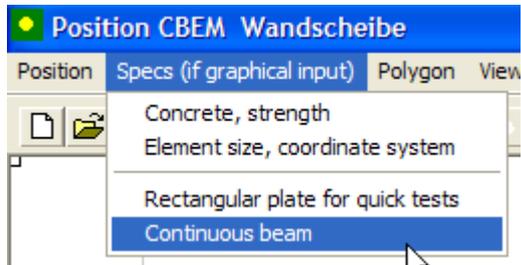
Fig 3.8 Plate attached to transverse supports

### 3.13. Continuous Beams

To model a continuous beam as a plate click on the menu entry

**Specs (if graphical input)**

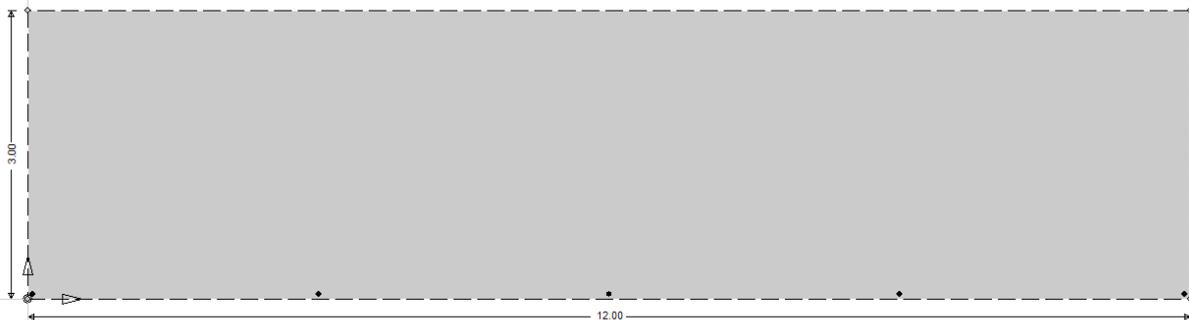
and choose the entry Continuous beam.



Next specify the dimensions of the plate and the number of spans

The image shows a dialog box titled 'Continuous beam'. It contains three input fields: 'Length' with the value '12', 'Width' with the value '3', and 'Number of spans' with the value '4'. There are 'OK' and 'Stop' buttons on the right side of the dialog.

The program will automatically generate a plate with the necessary number of point supports.

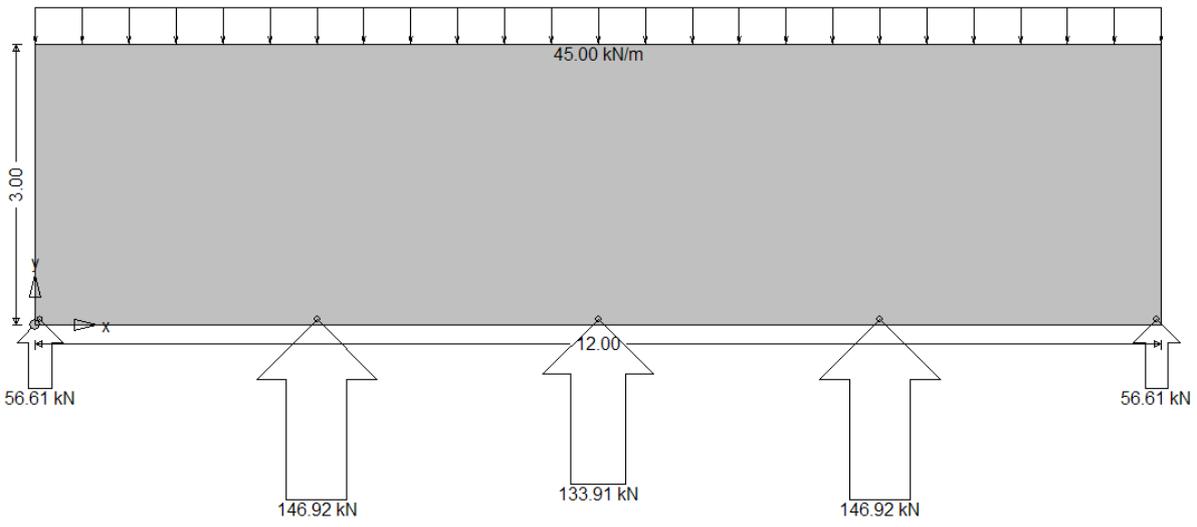
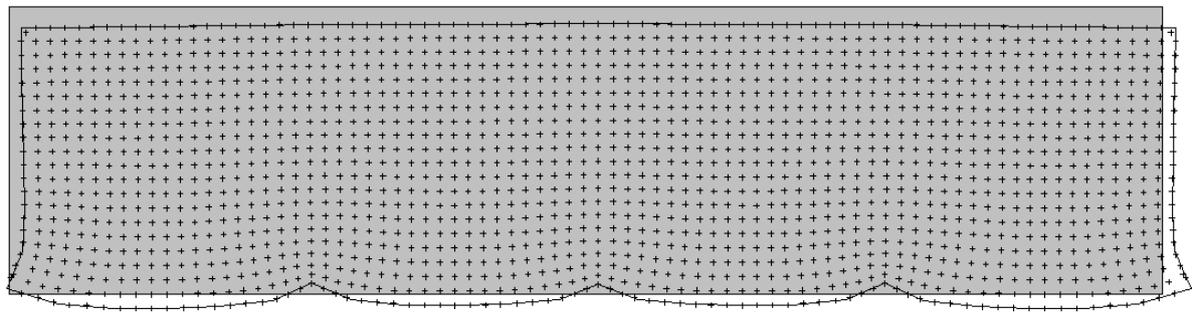
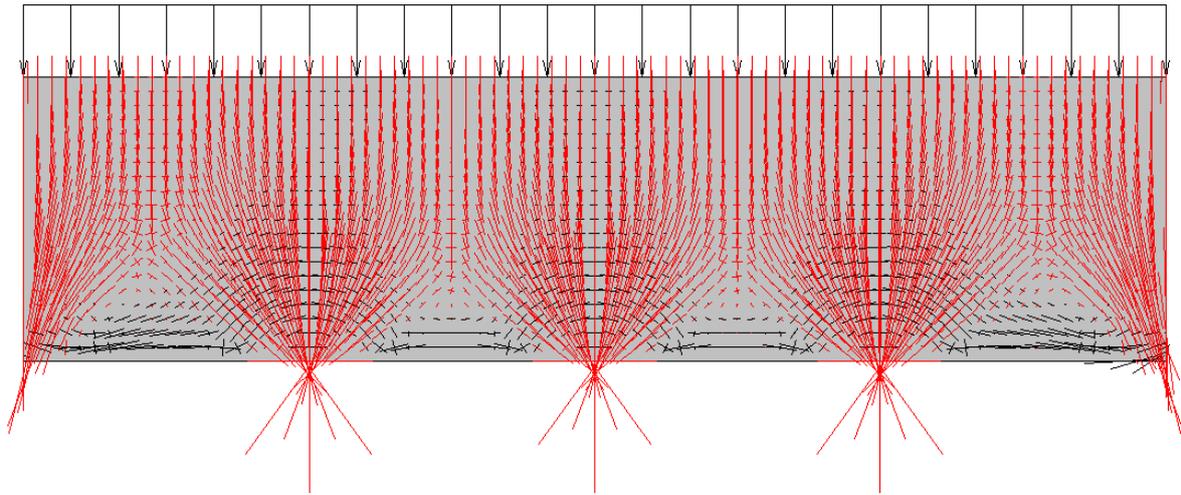


The point supports are placed inside the plate and their stiffness is set to very large values so to resemble fixed supports.

The image shows a dialog box titled 'Point supports (fixed points)'. It contains a table with the following data:

	x [m]	y [m]	H fixed	V fixed	cx [kN/m]	cy [kN/m]
1	0.05	0.05	Y	Y	10000000000.00	10000000000.00
2	3.00	0.05	N	Y	10000000000.00	10000000000.00
3	6.00	0.05	N	Y	10000000000.00	10000000000.00
4	9.00	0.05	N	Y	10000000000.00	10000000000.00
5	11.95	0.05	N	Y	10000000000.00	10000000000.00
6						

There are 'OK' and 'Stop' buttons on the right side of the dialog. A legend below the table indicates: H = horizontal, V = vertical, Y = Fixed, N = Not fixed.



## 4. LOAD

Possible loads are

- **gravity load**
- **line loads**
- **point loads**
- **partial area loads**
- **edge loads, edge displacements**
- **settlement of point supports**
- **dislocations at single points (to generate influence functions for stresses)**
- 



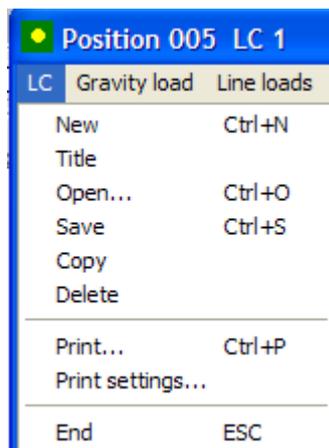
To enter a new load case click on the icon new



And to save the load case click on the icon



If you click on the menu entry LC you can copy a single load case from a previously calculated position.



### 4.1. The sign

In Mathematics all displacements and forces which point in the direction of the positive coordinate axes are positive.

But an engineer usually counts displacements and forces which point downward as positive even if he uses a type 1 system of coordinates where y points upwards. The program follows this rule.

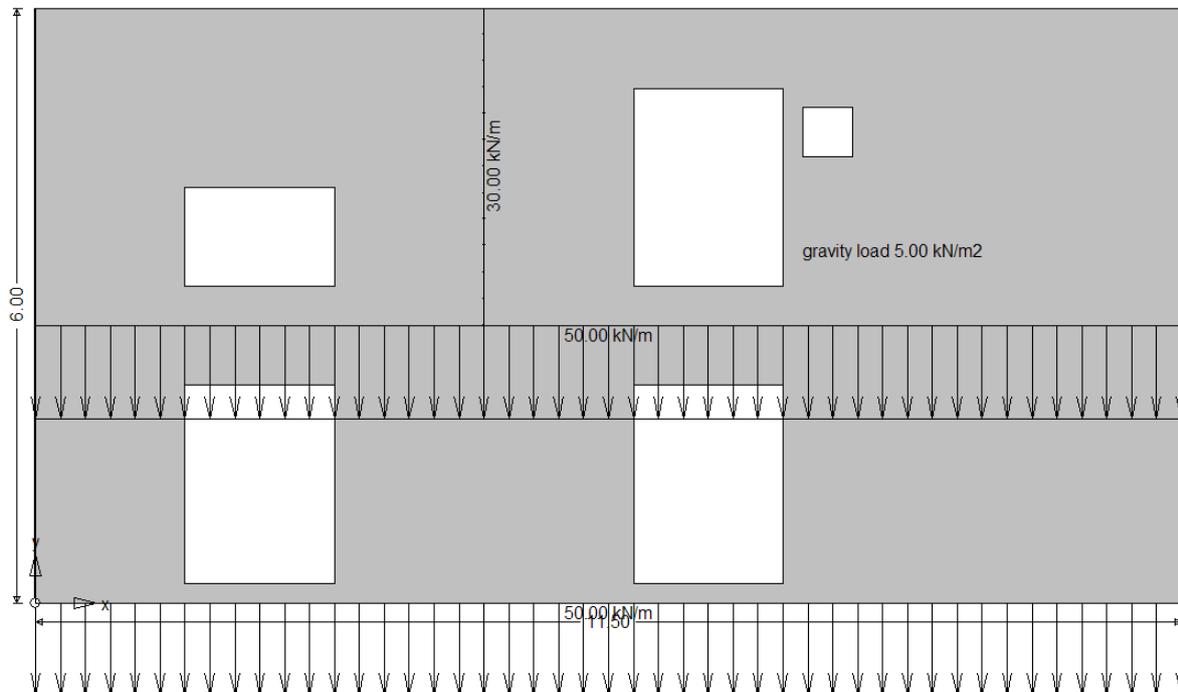
If you want to restore the mathematical order delete the file

SIGNUMV

in the program folder.

To reenact the engineering version copy a file with the name SIGNUMV into the program folder.

The contents of the file make no difference. Only the name is important.



Line loads									
	xa [m]	ya [m]	xb [m]	yb [m]	Value px_a	Value px_b	Value py_a	Value py_b	Comments
1	0.00	2.80	11.50	2.80	0.00	0.00	50.00	50.00	
2	4.50	2.80	4.50	6.00	0.00	0.00	30.00	30.00	
3									
4									

Edge loads								
	Edge	Side	rx a	rx b	ry a	ry b	Type	Comments
1	1	1	0.00	0.00	50.00	50.00	1	
2								

### 4.2. Gravity load

The dead weight of a concrete wall with a width of 12 cm is

$$g = \gamma * d = 25 \text{ kN/m}^3 * 0.12 \text{ m} = 3 \text{ kN/m}^2$$

and so the horizontal and vertical components of the corresponding area load are

$g_x = 0.0$                       horizontal component  
 $g_y = 3 \text{ kN/m}^2$                 vertical component

### 4.3. Line loads

Line loads [kN/m] have components in x- and y-direction.

They are specified by the position of the end points and the end values of the line load. In between the end points the distribution of the loads is linear. Line loads should act in the interior of the plate because they are modeled by the action of ,full space' kernels and not ,half-space' kernels of Boussinesq-type where the load acts at the edge of the half space.

Line loads that act on the edge of the plate should be entered as edge loads. To specify the location of a line load in graphics mode click on the icon



To use a dialog input click on the menu entry Line loads in the menu bar.

In graphical input mode you click on the (approximate position) of the end points of the line load and in the context menu that will open up you can specify the precise coordinates and the magnitude of the load.

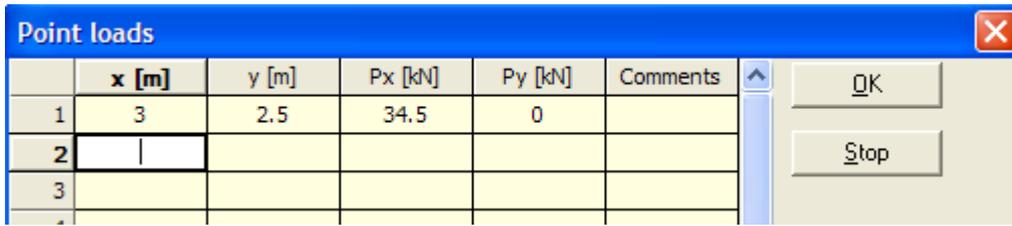
If you input additional line loads the dialog stays shut because the program assumes that the next line loads are copies of the first line load. To reactivate the dialog click on the right mouse button.

### 4.4. Point loads

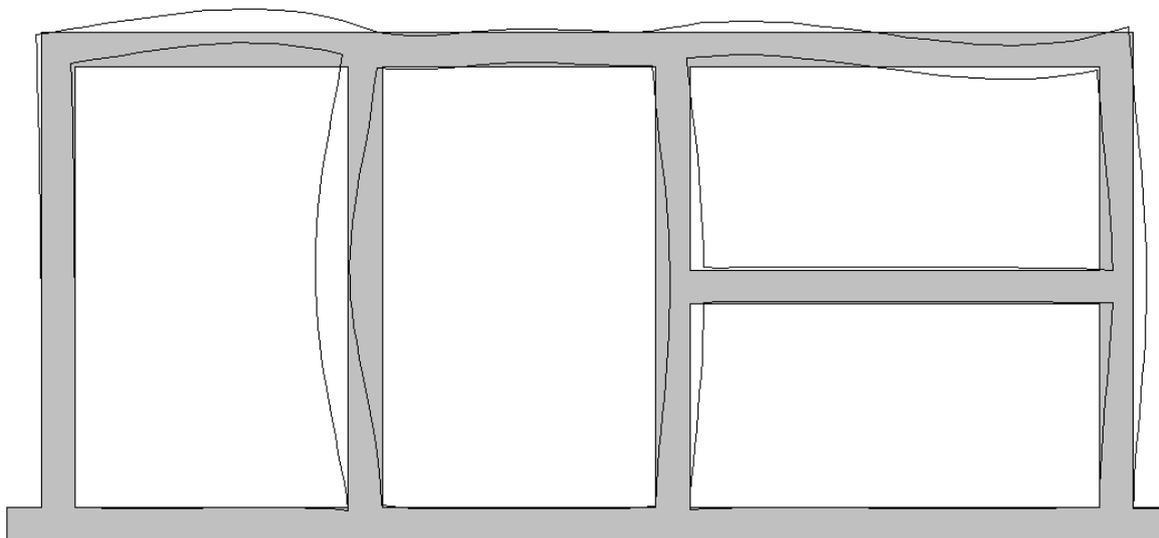
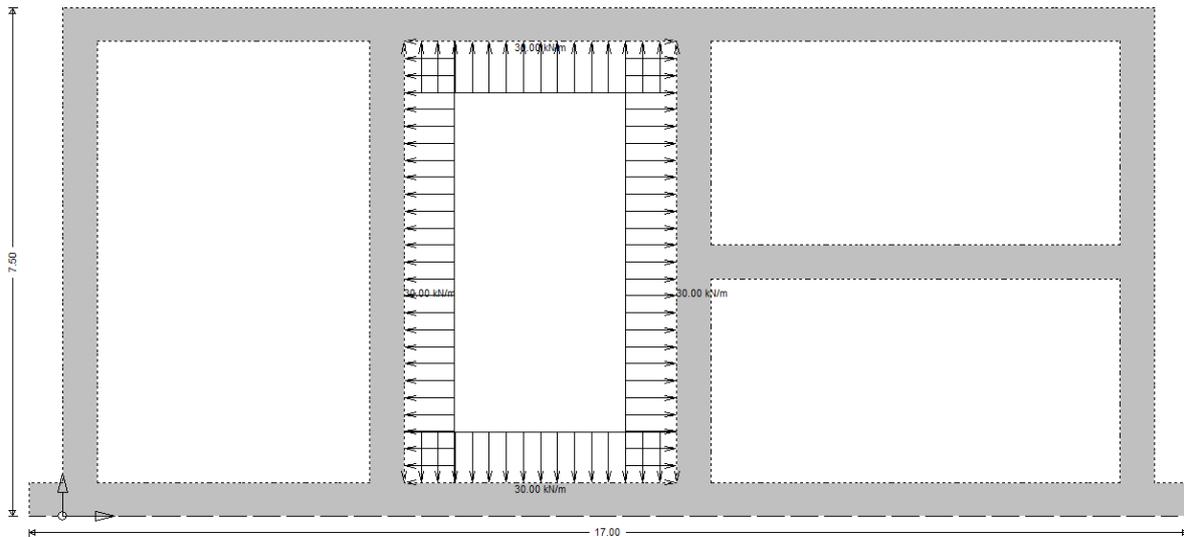
To activate graphical input mode click on the icon



and for dialog input mode click on the menu entry Point loads in the menu bar.



Point loads are modeled as 'full space' point loads so that the point of attack should lie inside the plate. Point loads on the edge of the plate should be moved inside the plate. A short displacement of, say, 0.1 m suffices.



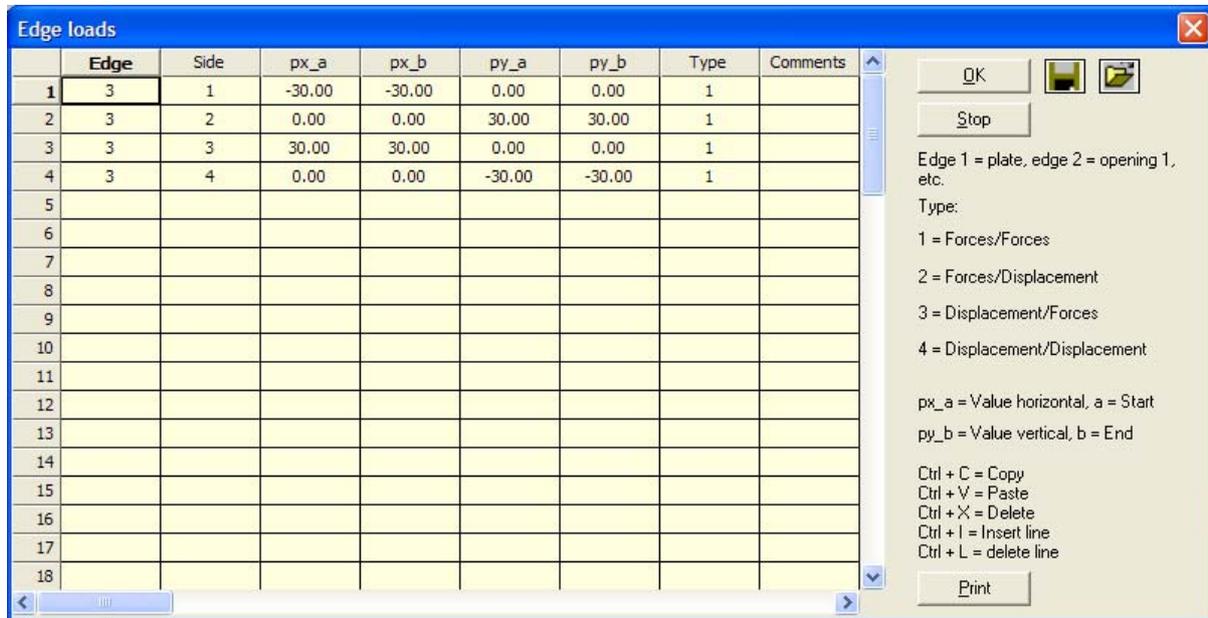
**4.5. Edge loads**

Edge loads are similar in nature to line loads. They act in horizontal and vertical direction.

But edge loads can also be displacements. Along the edge of a plate two force terms and two displacements are specified

$u_x$   $u_y$             displacements  
 $t_x$   $t_y$             tractions (edge stresses)

Any combination of two of these four boundary values can be specified.



The boundary values in horizontal direction are

$px_a$             at the start of the edge  
 $px_b$             at the endpoint

and likewise in vertical direction

$py_a$             at the start of the edge  
 $py_b$             at the endpoint

The value 1, 2, 3 or 4 in the column named Type specifies which combination of boundary terms these values represent.

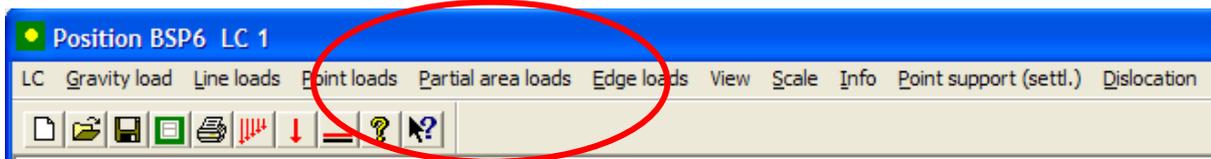
A plate with two openings has 3 = 1 + 2 edges, the outer edge plus the two edges of the openings. So you must specify at which edge the load is acting. The sides of an edge are the single segments which form the edge. Typically a rectangular plate has four sides. But you could also subdivide single sides into 3, 4 or more straight segments if you intend to apply edge loads to just some parts of a side.

In this case you must know beforehand where the partial edge load is applied and subdivide the edge of the plate accordingly in single pieces if necessary.

Edge loads can also be applied to elastic supports or tangential supports if the edge load acts in the unrestrained direction.

#### 4.6. Partial area loads

These are volume forces which act only in some portion of the plate. To start the dialog input mode of these loads click on the entry Partial area loads in the menu bar.



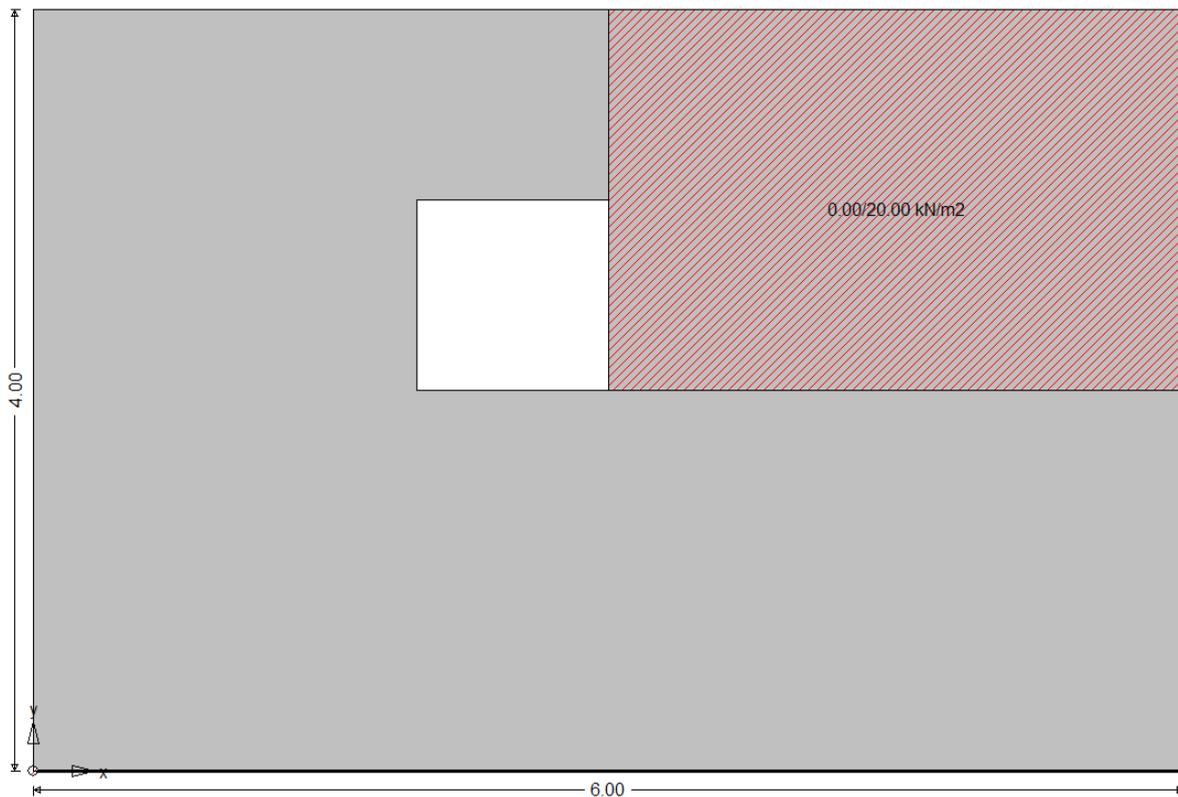
and specify the extension and magnitude of the load

Vertices of partial area loads

	x [m]	y [m]	px [kN/m <sup>2</sup> ]	py [kN/m <sup>2</sup> ]
1	3	2	0	20
2	6	2	0	20
3	6	4	0	20
4	3	4	0	20
5				

Buttons: OK, Stop

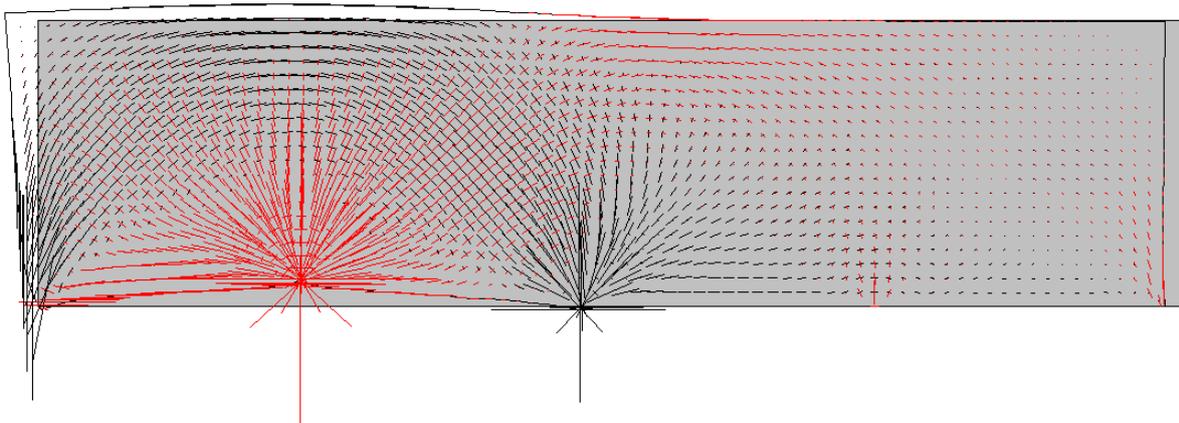
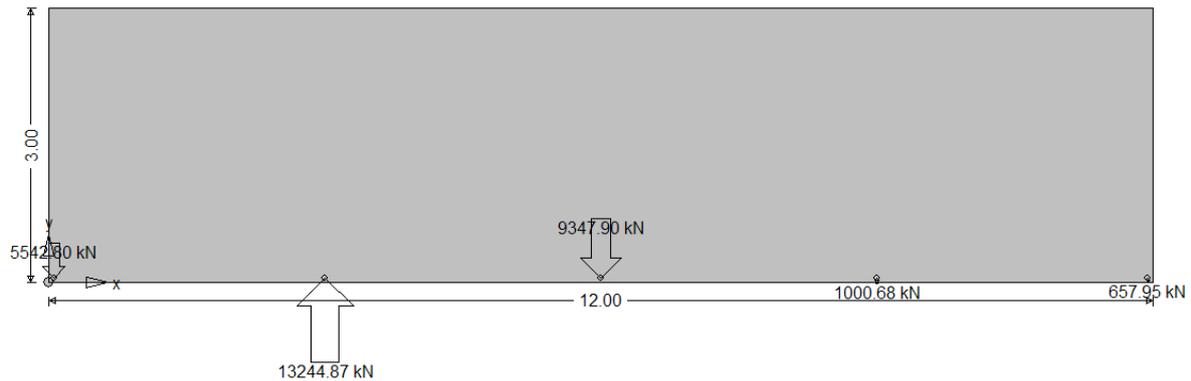
Separate multiple loads by a blank line!



**4.7. Settlements of point supports**

Point supports can be displaced in horizontal and vertical direction. To start click on the menu entry Point supp. (settlement) in the menu bar.

	x [m]	y [m]	u [mm]	v [mm]	Comments
1	0.05	0.05	0.00	0.00	
2	3.00	0.05	0.00	10	
3	6.00	0.05	0.00	0.00	
4	9.00	0.05	0.00	0.00	
5	11.95	0.05	0.00	0.00	
6					



From a theoretical point of view this load case does not exist because a single point of a plate can be pushed freely in any direction. Theoretically no force is necessary to displace a *single* point. The effects you observe are due to the approximate character of the numerical method.

On the other hand one could argue that the effects are approximately true if the point support is assumed to have a certain extend in horizontal direction.

#### 4.8. Dislocations

With dislocations you can calculate influence functions for stresses.

Dislocations at a point (Infl. functions (GF) for stresses) ✕

x [m]

y [m]

Horizontal dislocation [m] (1.0 = GF for sigma\_xx)

Vertical dislocation [m] (1.0 = GF for sigma\_yy)

Set the other component to zero, if GF are calculated.

No additional loads in this load case.

A horizontal dislocation produces the influence function for  $\sigma_{xx}$  and a vertical dislocation produces the influence function for  $\sigma_{yy}$ .

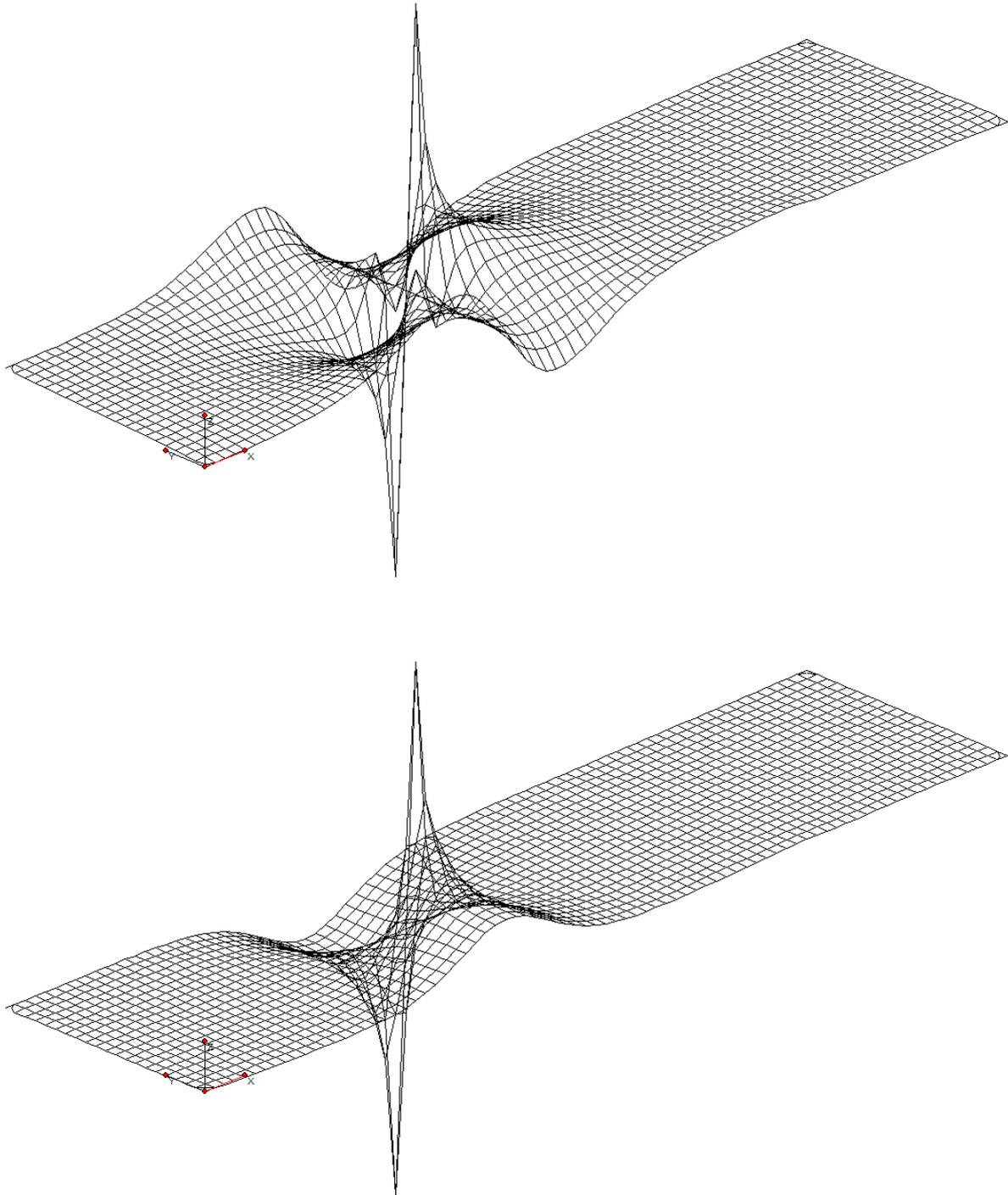


Fig. 4.1 Influence functions for  $\sigma_{xx}$ . Displayed are the horizontal (first picture) and vertical displacements caused by a unit dislocation in horizontal direction at the source point. The displacements are in-plane displacements so it would be more natural to use contour lines. But they are plotted here as out of plane values to better visualize the peculiar character of these functions.

## 5. STRESS POINTS

The stress points are the nodes, the Gauss points, so to speak, in a BE-program. At these points are calculated the stresses and the displacements. Because there is no mesh these points must be generated separately in a BE-program. Note that the accuracy does not depend on the mesh width that is the mesh (or grid of points) can have any shape, any size and any distance between the points

Stress points come in different sets.

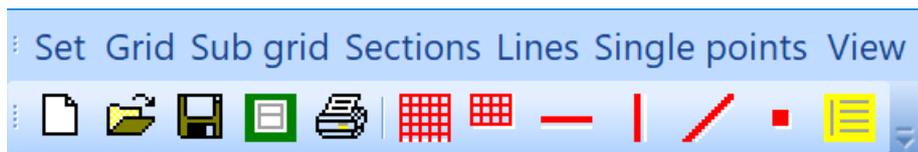
#1: The first set is usually a regular grid of stress points which covers the whole plate.

#2: Set number two may consist of a set of single lines of stress points which allow a closer inspection of certain regions of a plate.

#3: Set number three may consist of single stress points where additional information about the stress field is needed.

To start the input of the stress points click on the entry **P**oints in the main menu

### 5.1. 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



### 5.2. Grid

By clicking on the icon



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

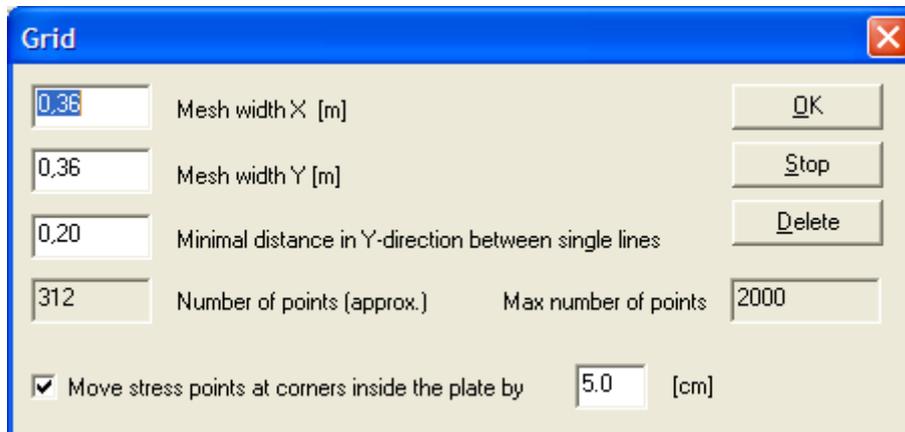


you can generate a sub grid, which only covers a part of the plate. If the icon of the sub grid is pressed down you can draw the shape of the sub grid with the mouse. When you return to the starting point a dialog will pop up where you can specify additional details.

Alternatively you can specify the shape of the sub grid in dialog input mode by clicking on the entry

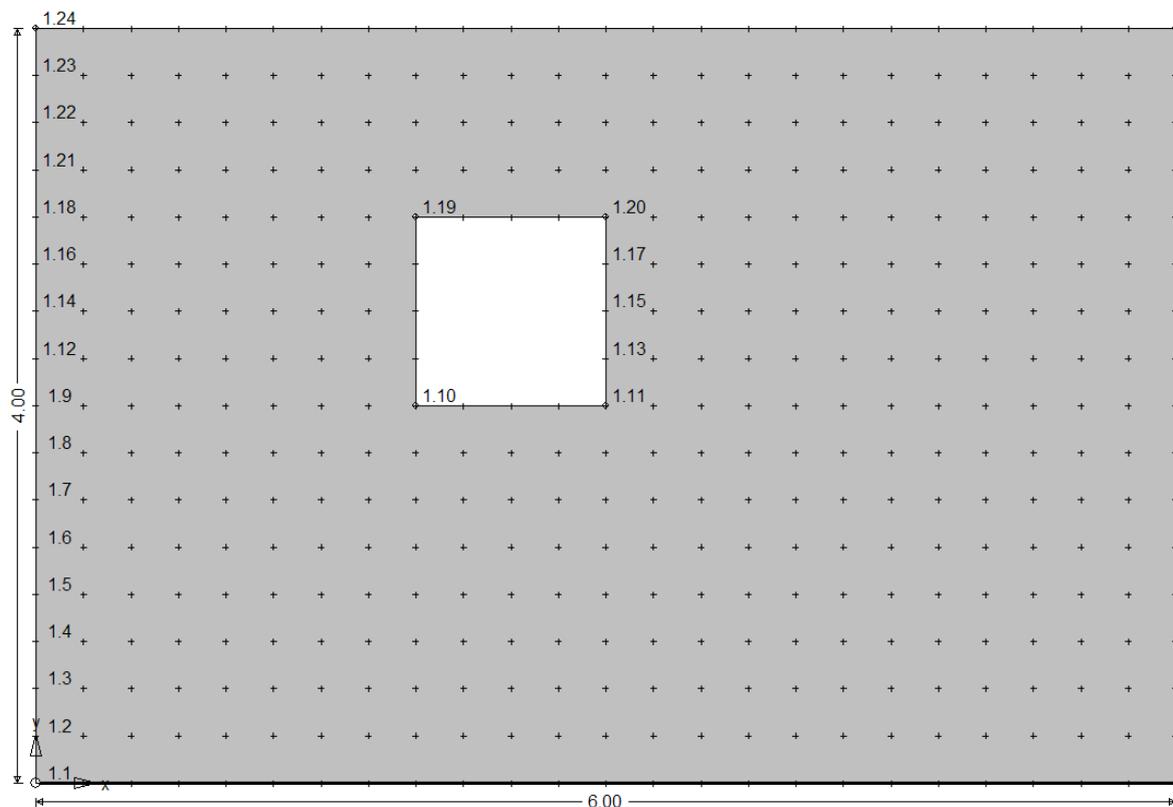
**Sub grid** in the menu bar.

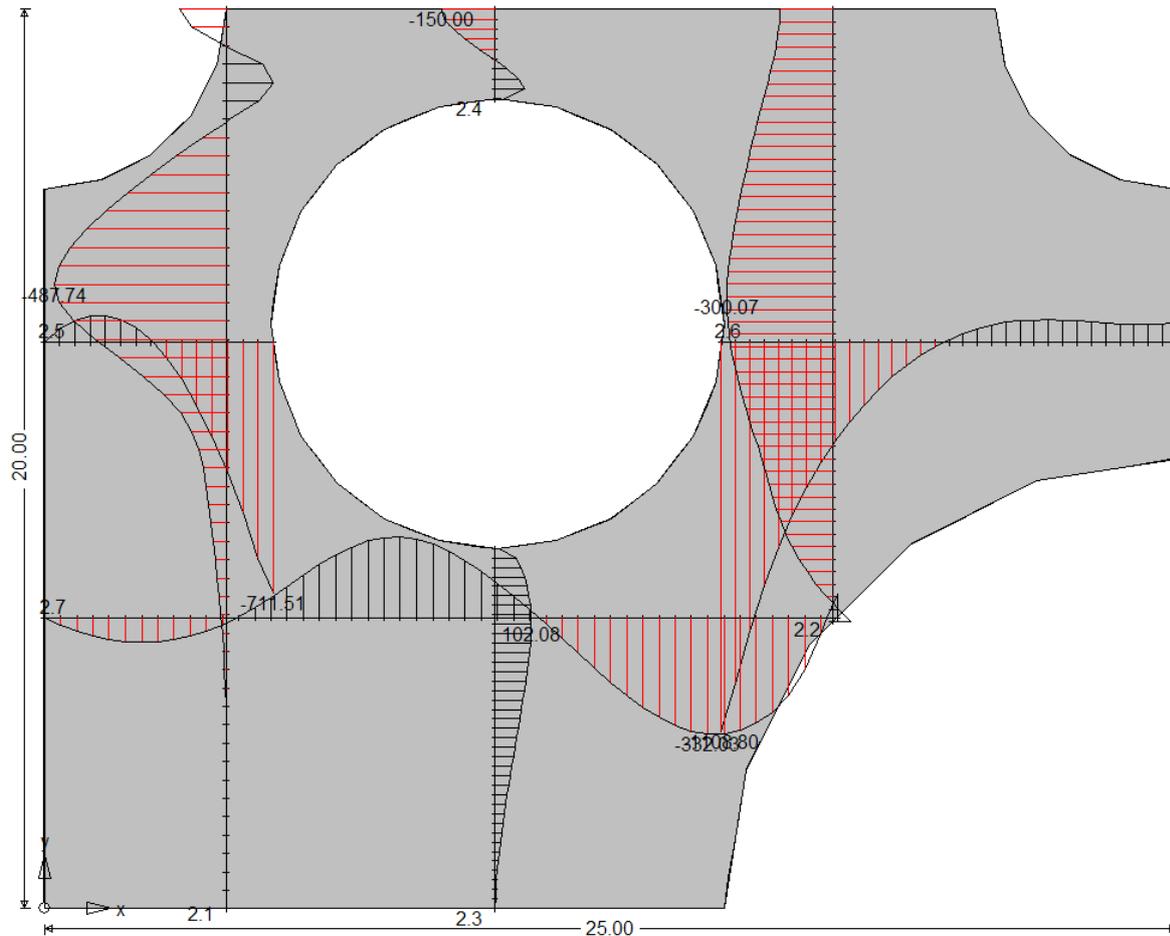
Both types of grids (meshes), the global grid and the sub grid, are determined by the mesh width in horizontal and vertical direction. The grids are actually not a mesh but rather a set of horizontal lines on which the points are distributed in (approximately) equal distances.



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.





Stress points which happen to lie at corner points are ,critical' in that the stresses at such points are often ambiguous because of changing boundary conditions and/or the change in the direction of the normal vector on the boundary. To avoid that results at such points spoil the global picture you can let the program move these points inside the plate by a certain distance.

### 5.3. Sections

To specify results along certain cross sections of the plate you define horizontal or vertical sections either interactively by clicking on the icons



or you can enter them manually by clicking on the entry **Sections** in the menu bar.

### 5.4. Lines

Single stress lines in an arbitrary direction and with arbitrarily positioned end points can be specified in graphical input mode with the mouse



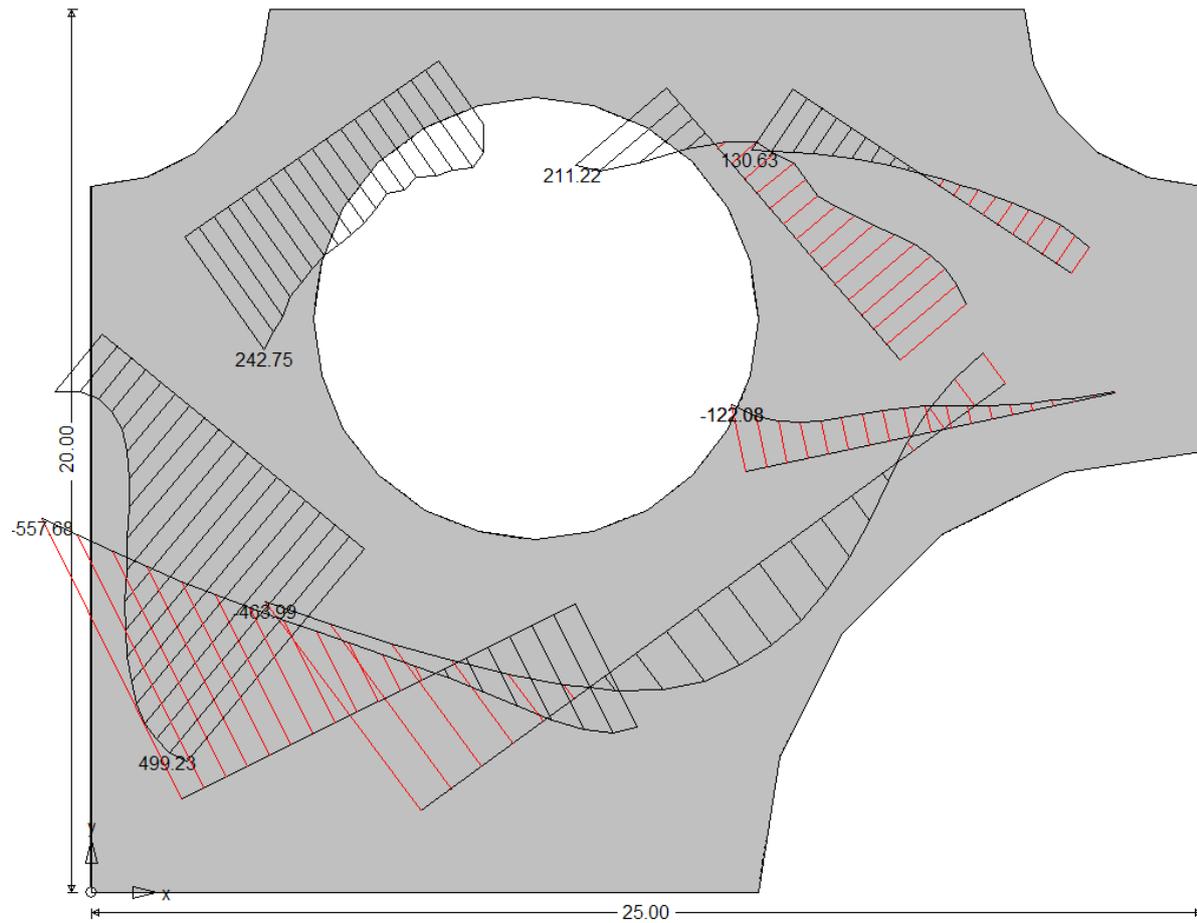
or in dialog input mode by clicking on the entry **Lines** in the menu bar.

### 5.5. Points

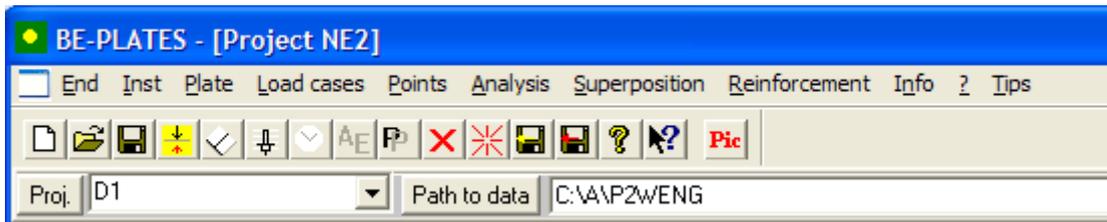
To specify single stress points either activate graphical input mode by clicking on



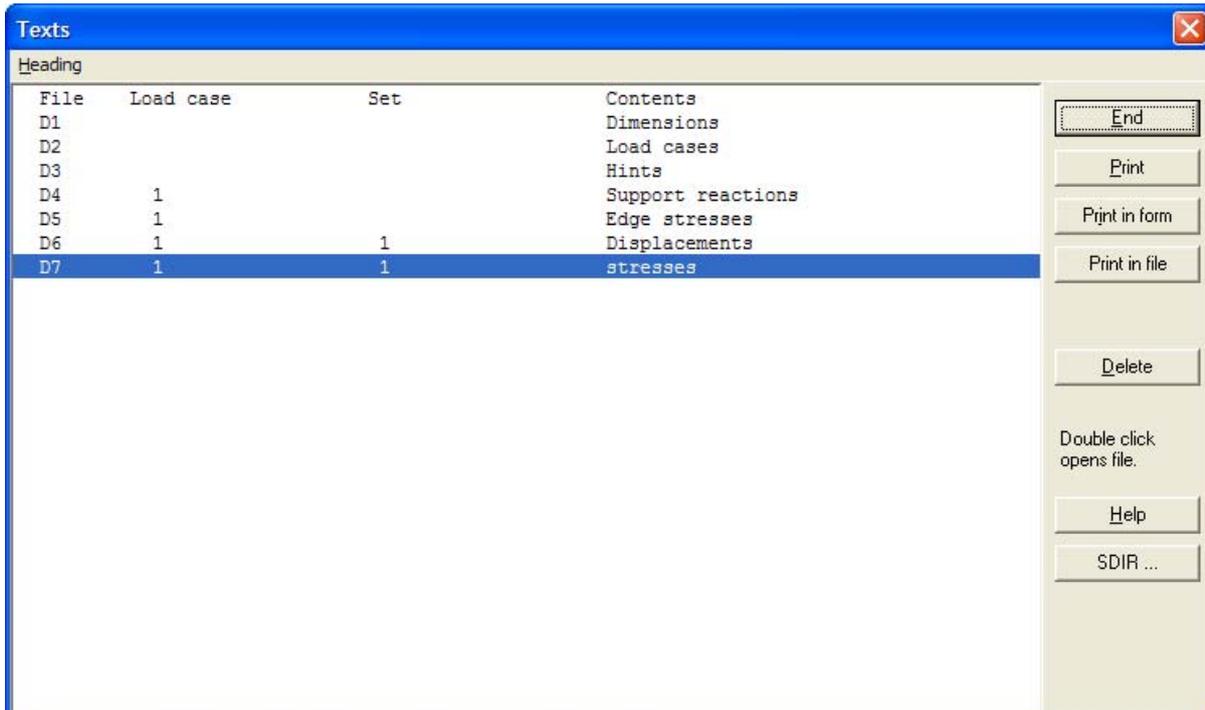
or input them manually by clicking on the entry **Points** in the menu bar.



## 6. RESULTS



The results of a plate analysis are stored in text files D3, D4, D5, etc.



You get to this dialog by clicking on the icon



For a graphical display of the results click on the icon



### 6.1. 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 discretization itself should be checked.

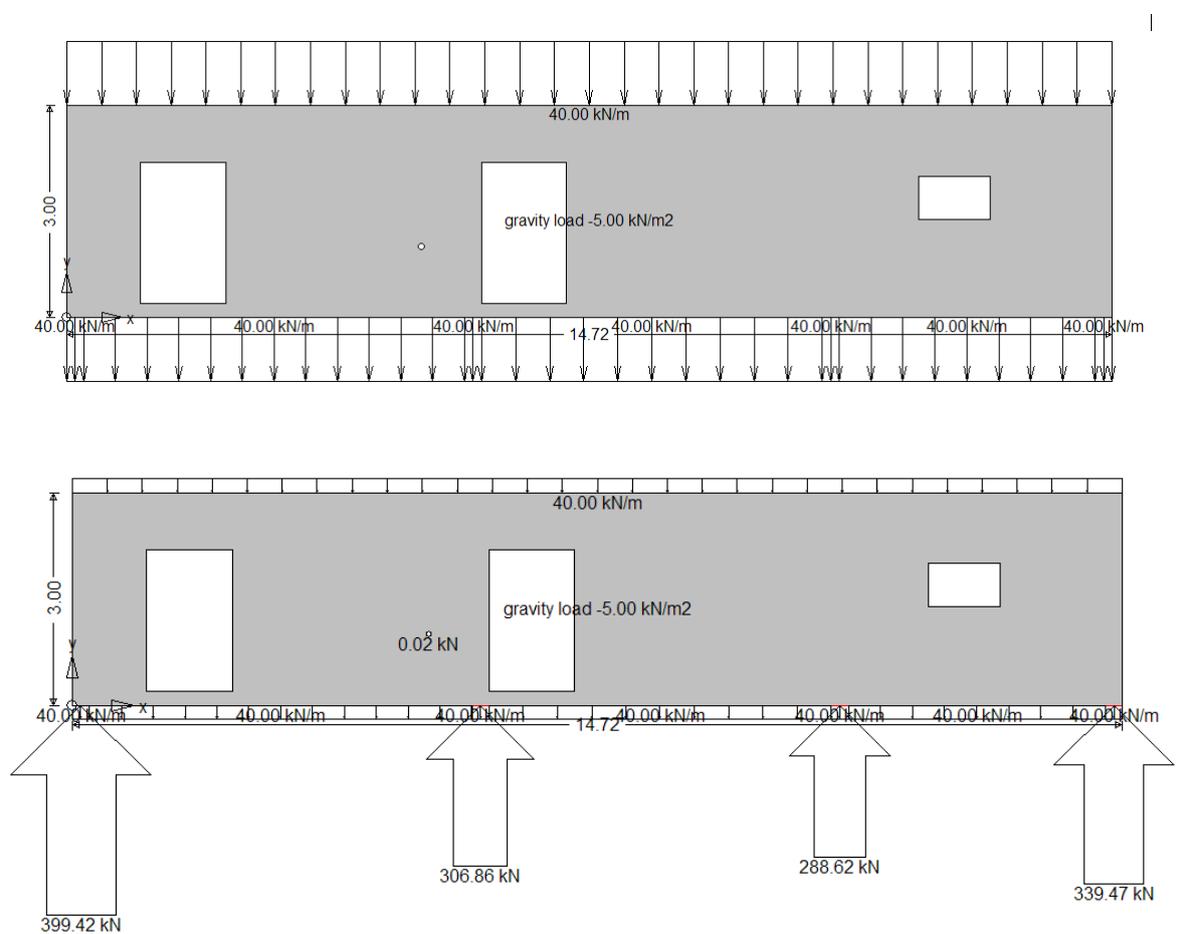
*Remark:* In the finite element method as in the boundary element method the programs solve an approximate load case exactly and the support reactions maintain equilibrium with regard to the loads of these

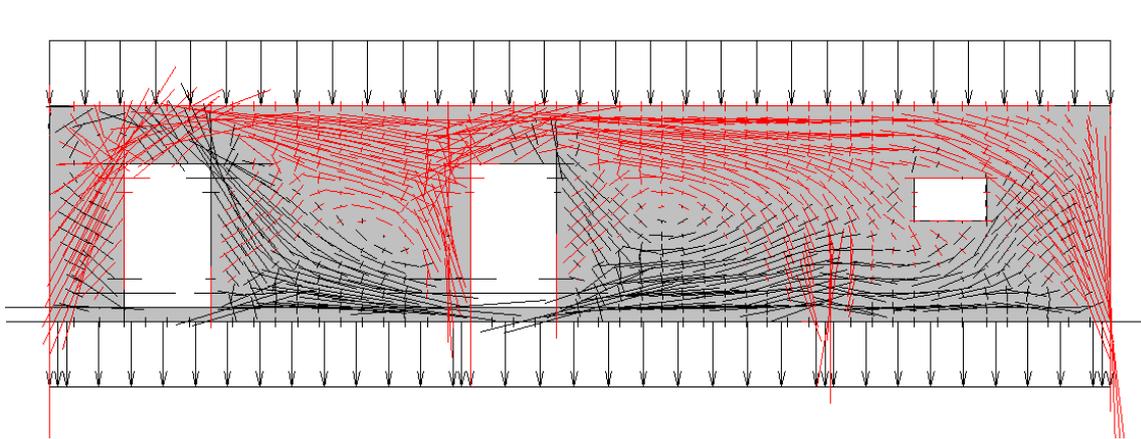
approximate load cases. So a discrepancy of 1% signals that the total load in the approximate load case differs by 1% from the total load of the original load case.

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*, 2<sup>nd</sup> Ed., Springer-Verlag, p. 184





## 6.2. Internal actions

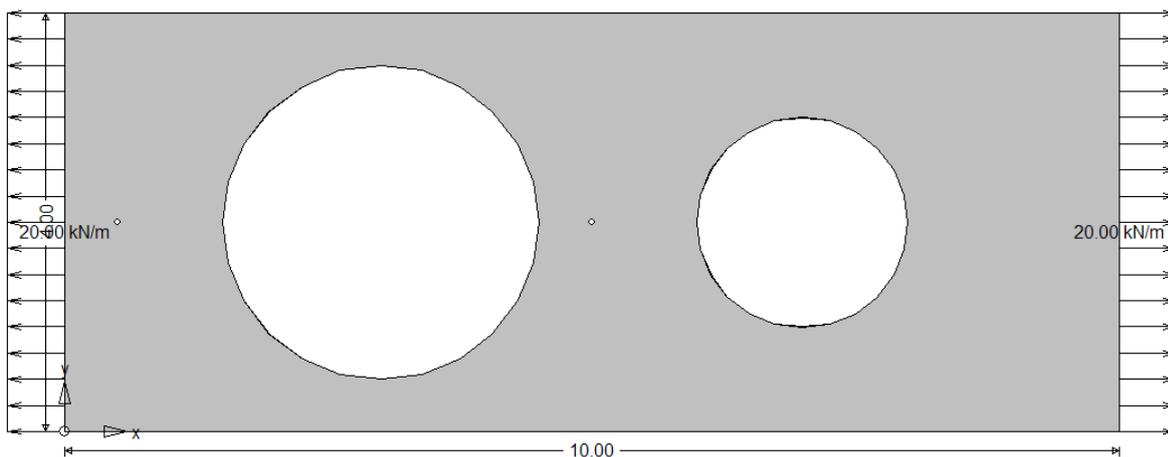
The program calculates at each stress point the displacements and stresses.

The stress points form a grid which consists of a series of horizontal lines on which are distributed the stress points. Along each line the program calculates also the resulting normal force  $N$ , the shear force  $V$  and the bending moment  $M$  if the line is thought of as a cut which passes through the plate.

## 6.3. Boundary values

The program outputs the following boundary values:

- $u_x$  horizontal displacement
- $u_y$  vertical displacement
- $t_x$  horizontal component of the traction vector  $\mathbf{t}$
- $t_y$  vertical component of the traction vector  $\mathbf{t}$



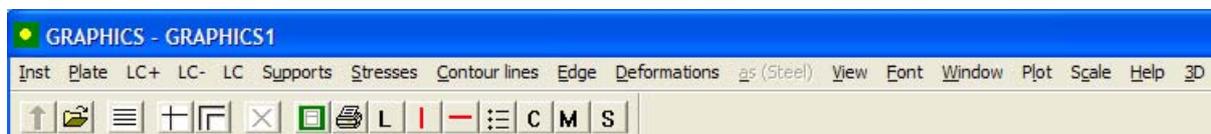


## 7. Display of results



To display the results click on the button Graphics.

This will start the program GRAPHICS



where the results of the analysis can be displayed as

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

### 7.1. 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).

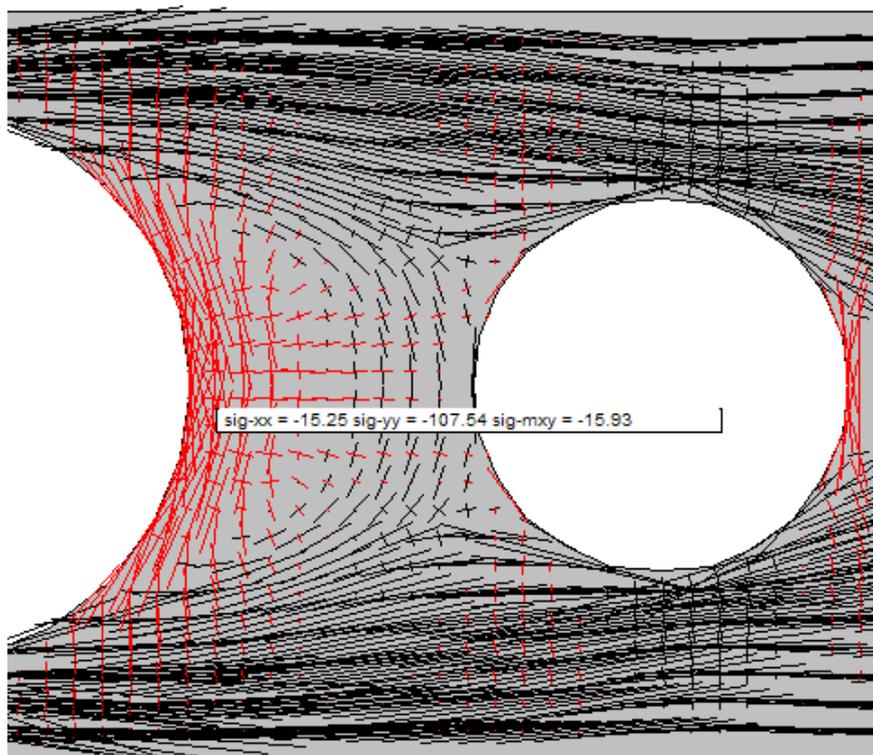
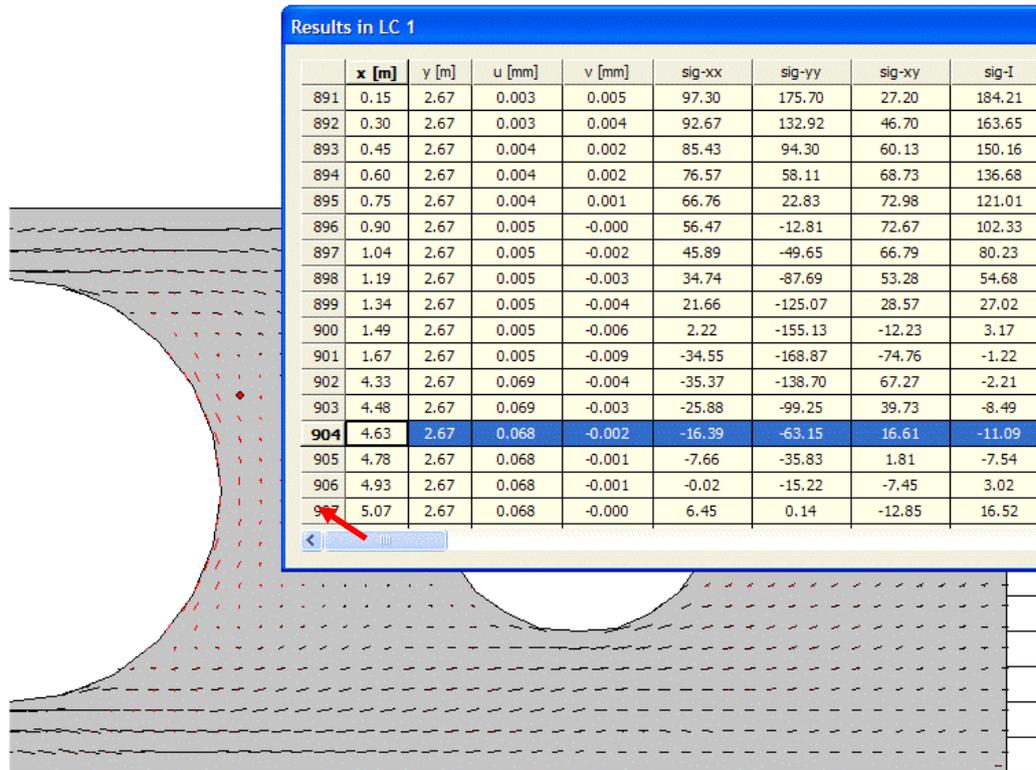
Click on the green edged printer icon for a print preview.

**Context menus**

In many windows a click on the right mouse button will open up 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  $x = \dots$ ,  $y = \dots$ , will be displayed on the screen with a small red dot.

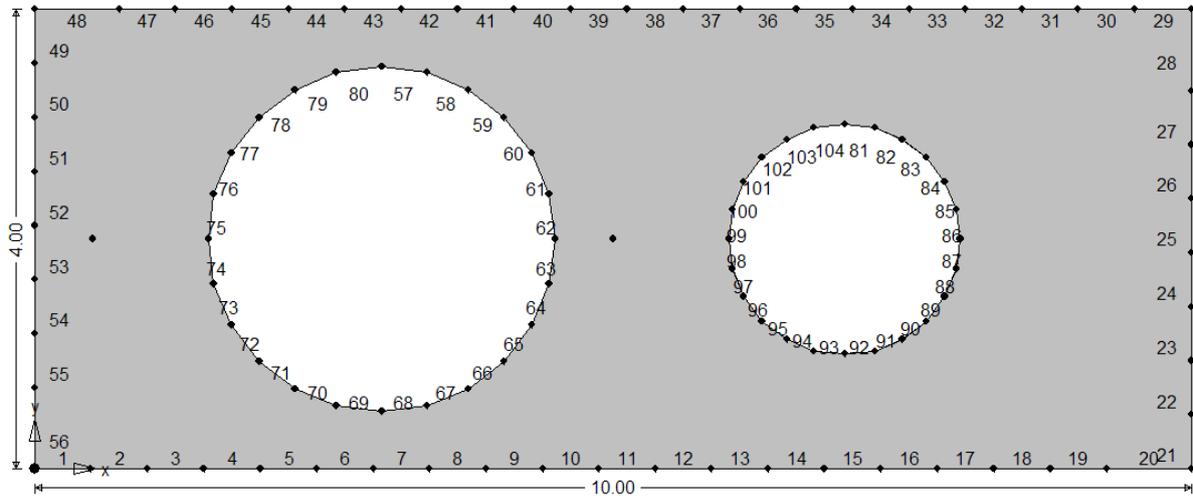


If you click on the principle moments diagram the moments of the nearest stress points are displayed on the screen.

### 7.1.1. Boundary elements

To display the plate and the (quadratic) boundary elements click on the entry **Elevation** in the main menu and choose

Details -> Elements

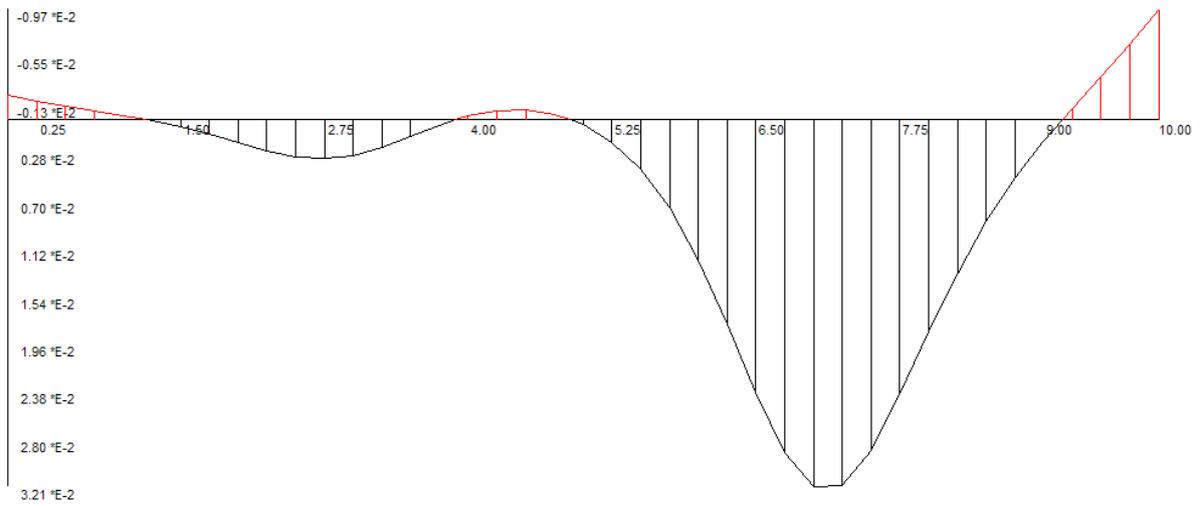


### 7.1.2. Boundary values

Click on Edge -> Boundary values

Boundary values	LC+	LC-	LC	Max	Min	Info
Support reaction H (x-component of stress vector)						
Support reaction V (y-component of stress vector)						
<hr/>						
Stresses xx						
Stresses yy						
Stresses xy						
<hr/>						
Horizontal displacement ( $u_x$ )						
Vertical displacement ( $u_y$ )						

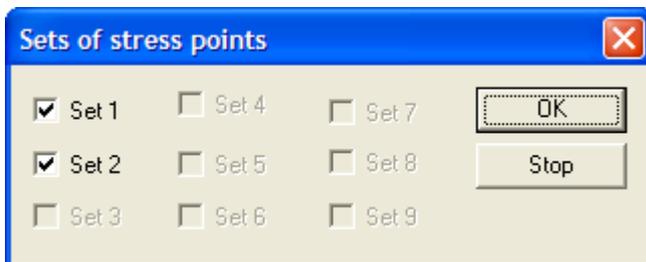
Choose a boundary value and click on a side to display the function.

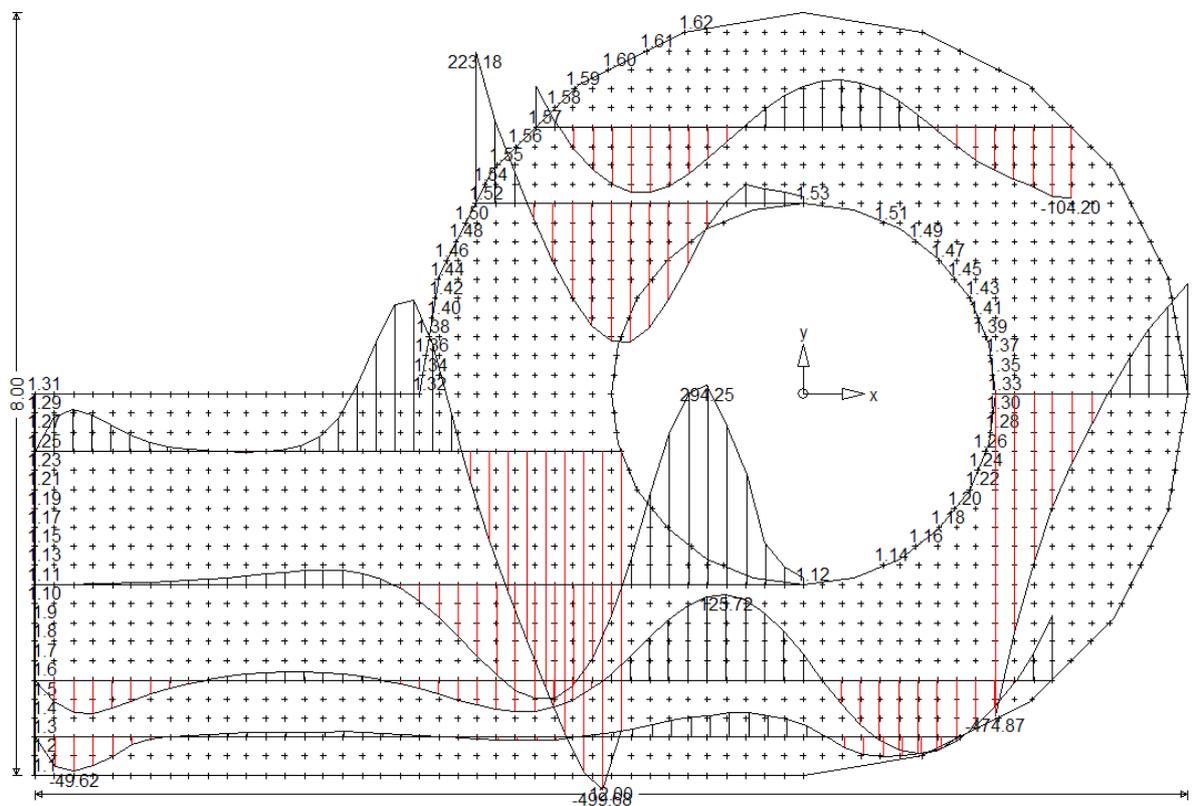
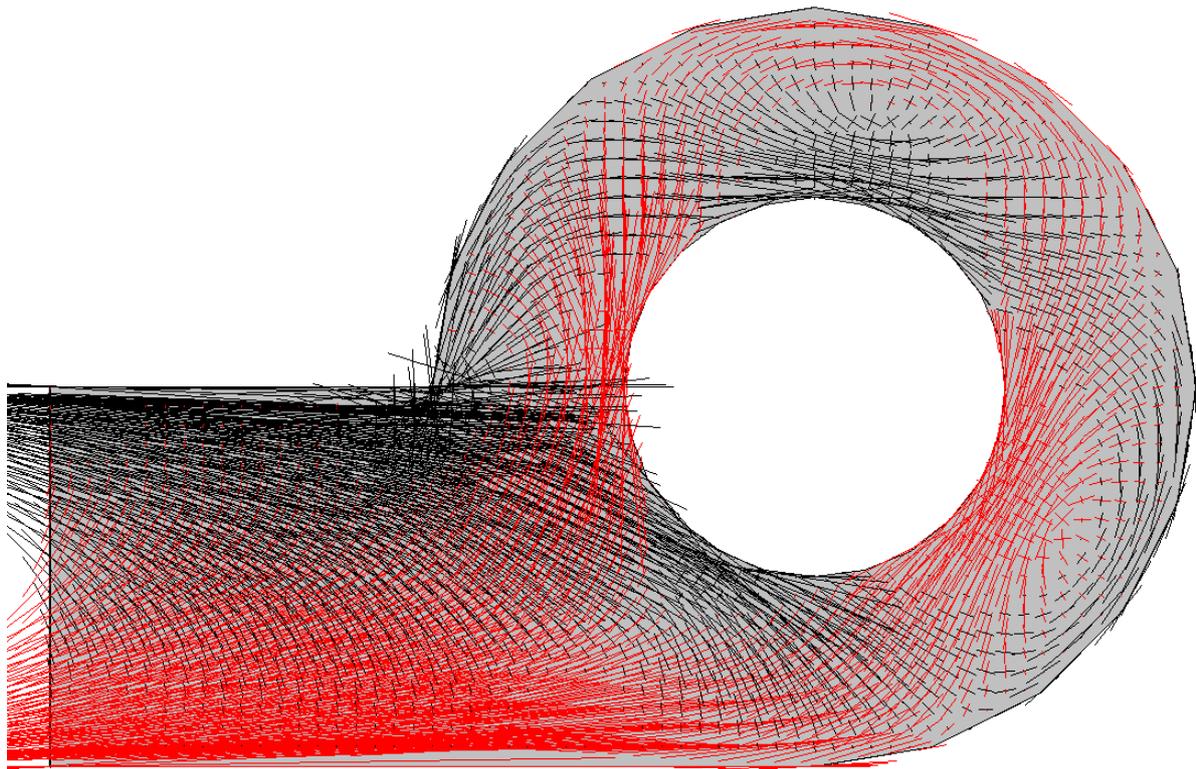


### 7.1.3. Stresses and displacements

To display the principle stresses click on the entry Stresses in the main menu. The program will display the principle stresses at all the single stress points.

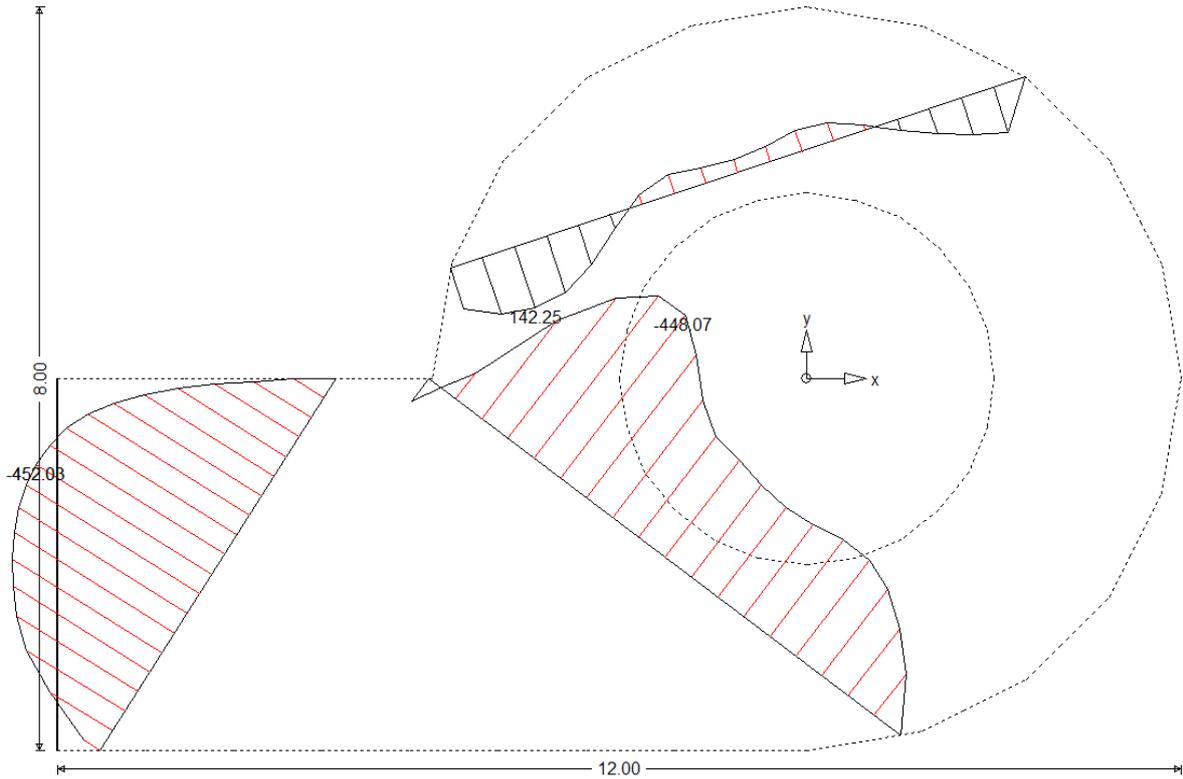
To restrict the display to only one particular set click on the entry **Choice of Stress points**



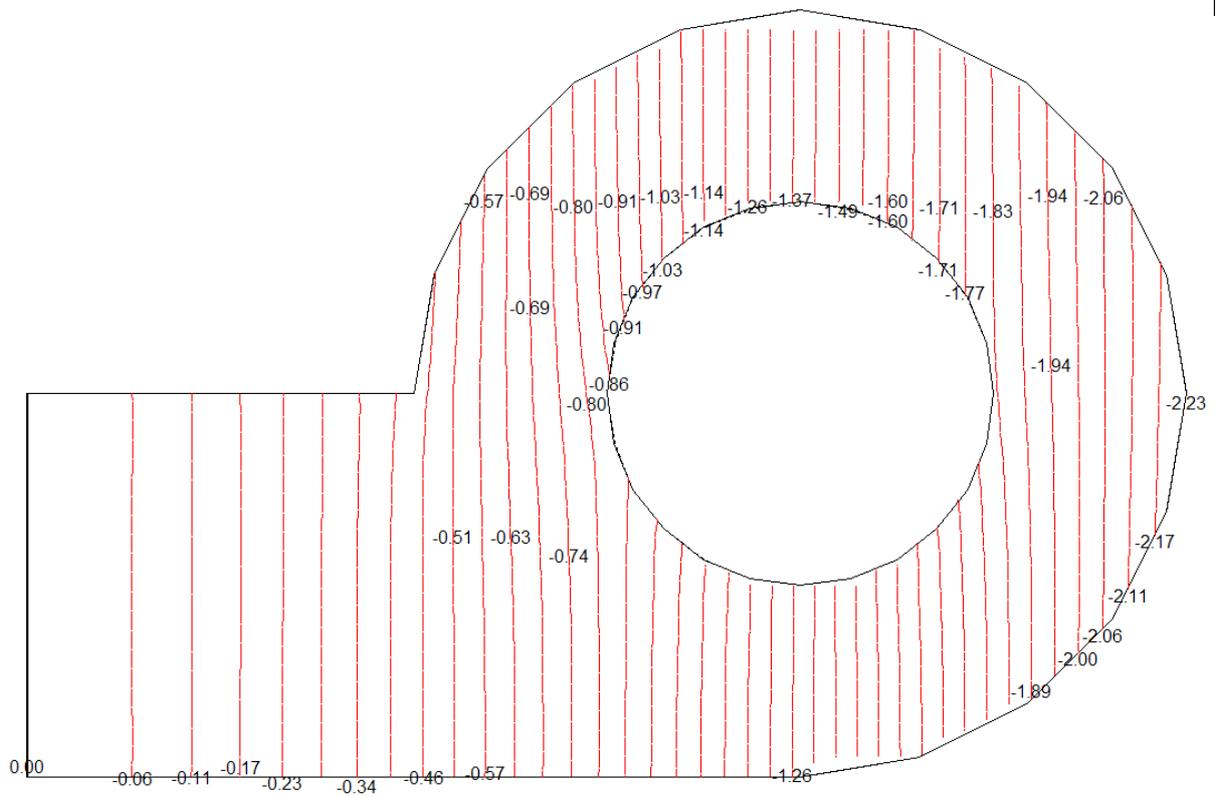


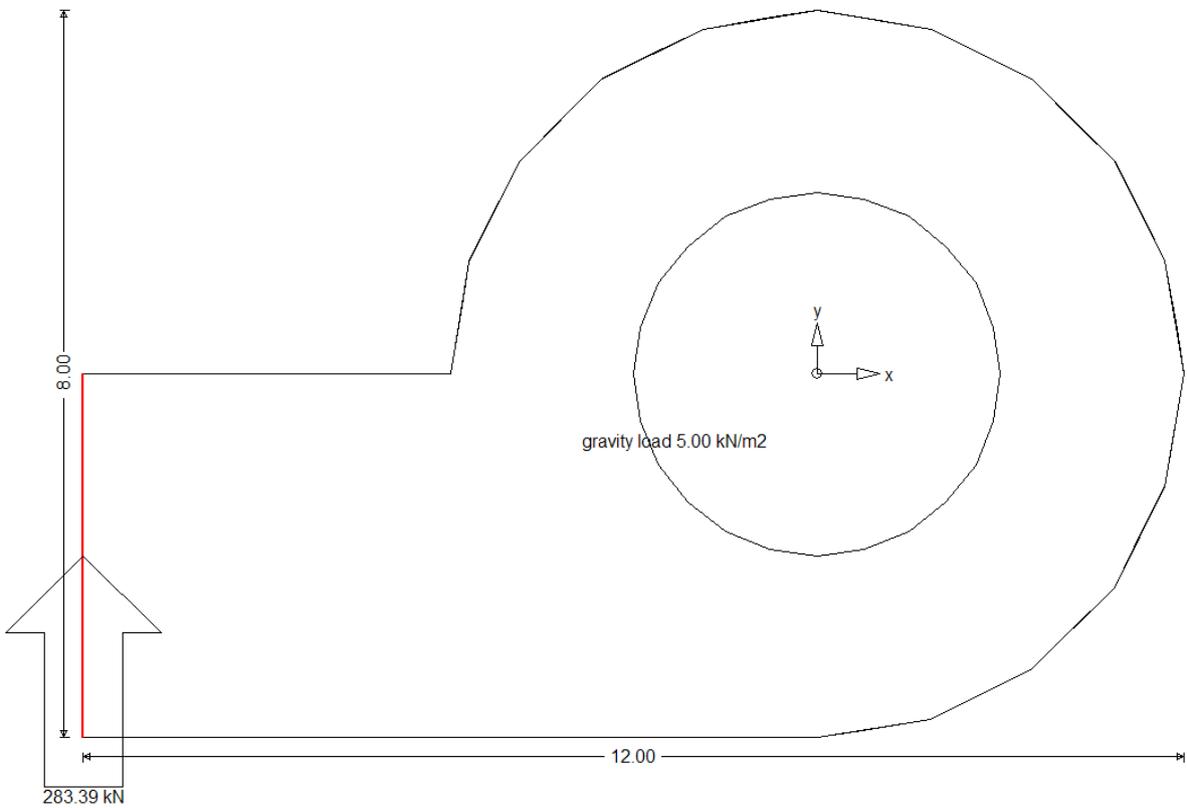
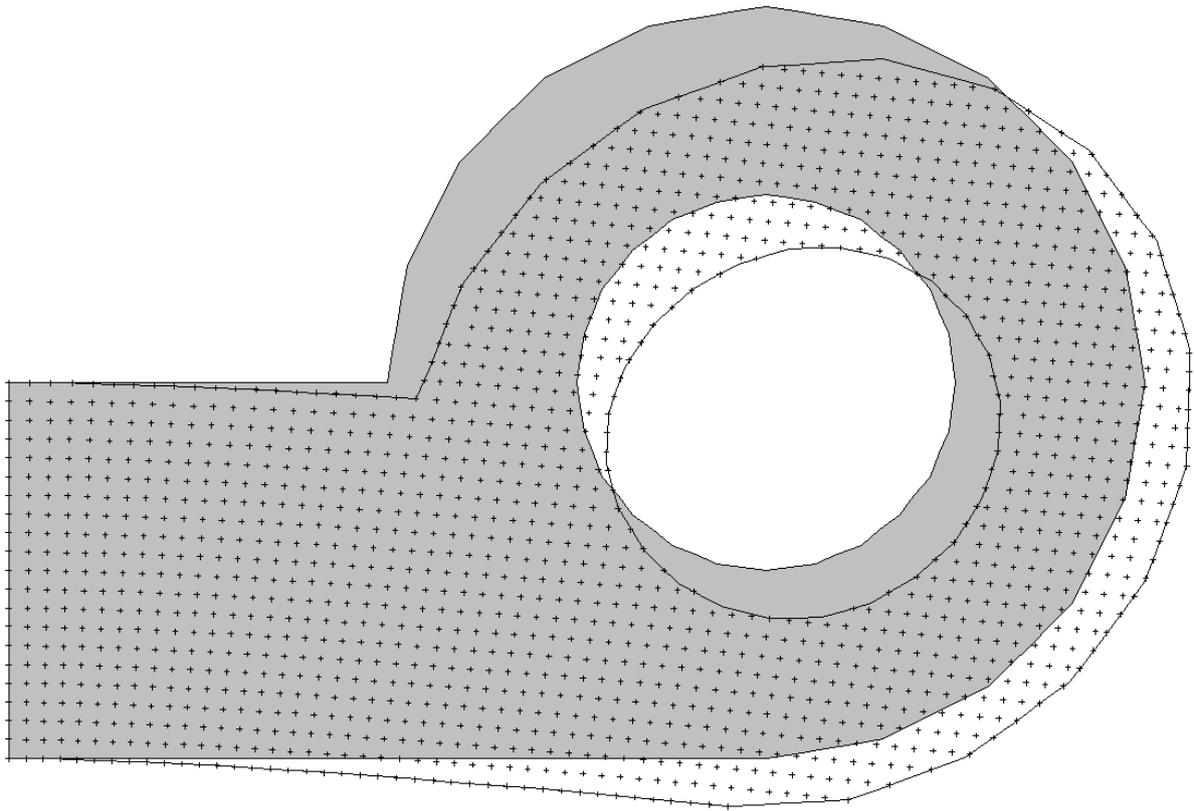
You can also click on single stress lines and display the results along such a line.

Or you can draw a line in arbitrary direction across the plate and display the results along such a line. To activate this option click on **Free Hand**

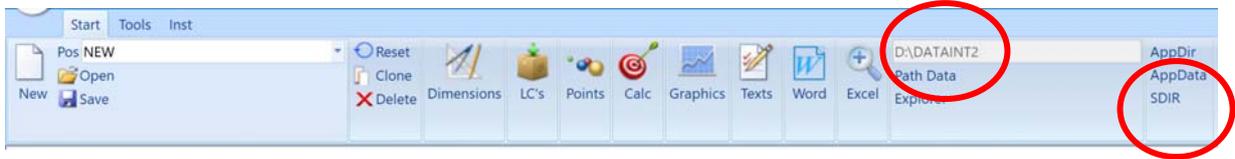


or display the result in the form of contour lines.





## 8. STORING AND RETRIEVING



### 8.1. Storing a complete position

The program stores the files of a project, 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 have to manually store the whole subdirectory on an external drive or on a CD.

### 8.2. Storing only the input

Otherwise you can only store the input of a position. This will allow you to recover 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.



After clicking on the folder 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
```

### 8.3. Retrieving a position

To retrieve a stored position, say position ABC, from the ACAD-files open the Windows-Explorer find the file ACADGEO.ABC and drag this file onto the open window of the program BE-PLATES.

This will automatically regenerate the complete position from the three ACAD-files. You only have to click on the Calc button to start the analysis.

This also works with the echo-files. When you have manually saved the ECHO-files of a position XXX then you can reinstall the position by dragging the file ECHO.XXX onto the program window.

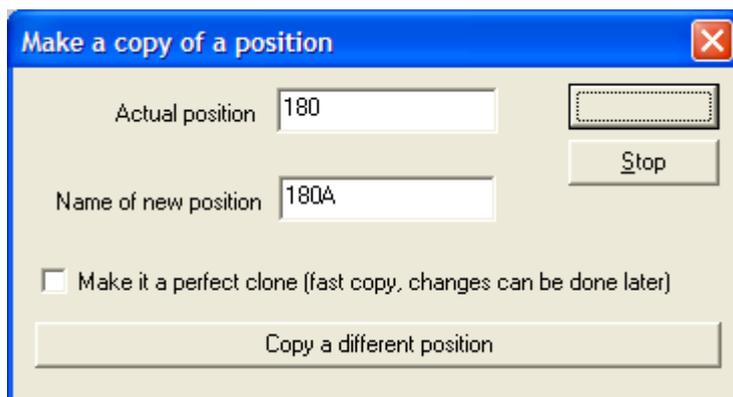
The contents of the ACAD- and ECHO-files are identical.

### 8.4. 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 slab on the screen allowing you to make modifications of the newly generated copy of the original slab.

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.

### 8.5. Resetting a position

A click on the icon



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

A click on the icon



will (physically) erase the position that is the folder that contains the files of the position will be deleted.