BE-LAPLACE

Manual

FEBRUARY 07, 2020

PROF. DR.-ING. FRIEDEL HARTMANN
HAGEBUTTENWEG 12
D-34225 BAUNATAL

TEL. +49-561-949-1510
FAX. +49-561-949-1511
WWW.BE-STATIK.DE

1. INSTALLATION

1. Unpack the file BE-LAPLACE.ZIP and click on the file setup.exe

2. The program is installed in the folder

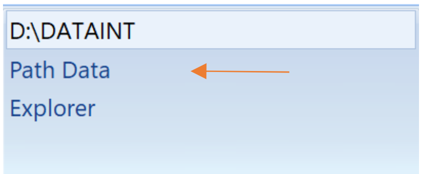
   C:\Programs (x86)\BE-STATIK\BE-LAPLACE 20 (or a higher version number than 12)

3. The program is written in C++ and compiled with Visual Studio 2010 and the MFC extensions. In case the standard dll libraries of Visual Studio 2010 are not installed on the computer (which is very rare) you need to install the dll's manually. Start the Windows-Explorer and open the program folder (see above) and do a double click on the file vcredist_x86.exe. This will install the missing dlls.
   Simple test: if the program starts the dll's are installed on the computer.

4. The program uses a Data Path. The Data Path is the name of a folder on your computer which you specify. It serves as the 'root folder' for the subdirectories SDIR… which contain the data files of the single positions. After the installation, the Data Path points to the directory

   C:\Users\...\AppData\Roaming\BE- LAPLACE 20

   The program cannot write to this folder to store your input. So, the first thing you need to do is to specify a working Data Path. For this click on the button Path Data

   and choose an existing folder as your data path or generate a new folder to which the program has write access.

5. The manual is included. Click on the tab Inst and then on Manual.

1.1. STARTING THE PROGRAM

To start click on the icon

BE-LAPLACE

on the desktop.

1.2. THE MAIN MENU

To begin a new problem, click on the button New and enter the name of the position, (or job), as for example 123 or AB. Next specify the shape of the domain by clicking on

Dimensions // shape of the domain

then on

LC's // loads

then on

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

and finally start the calculation by clicking on the button

Calc // **analysis of the problem**

The stress points are the points in the interior at which the program outputs the solution.

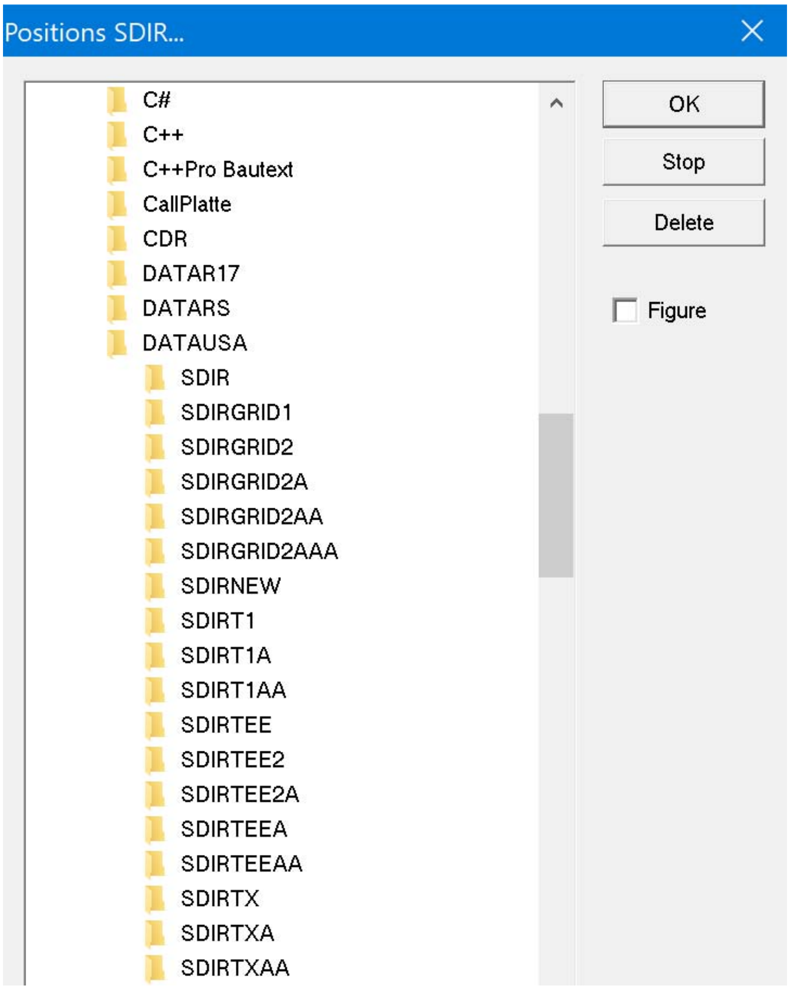
1.3. POSITIONS

Each problem, each job, 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 **SDIR** in the main menu will display the contents of the subdirectory.

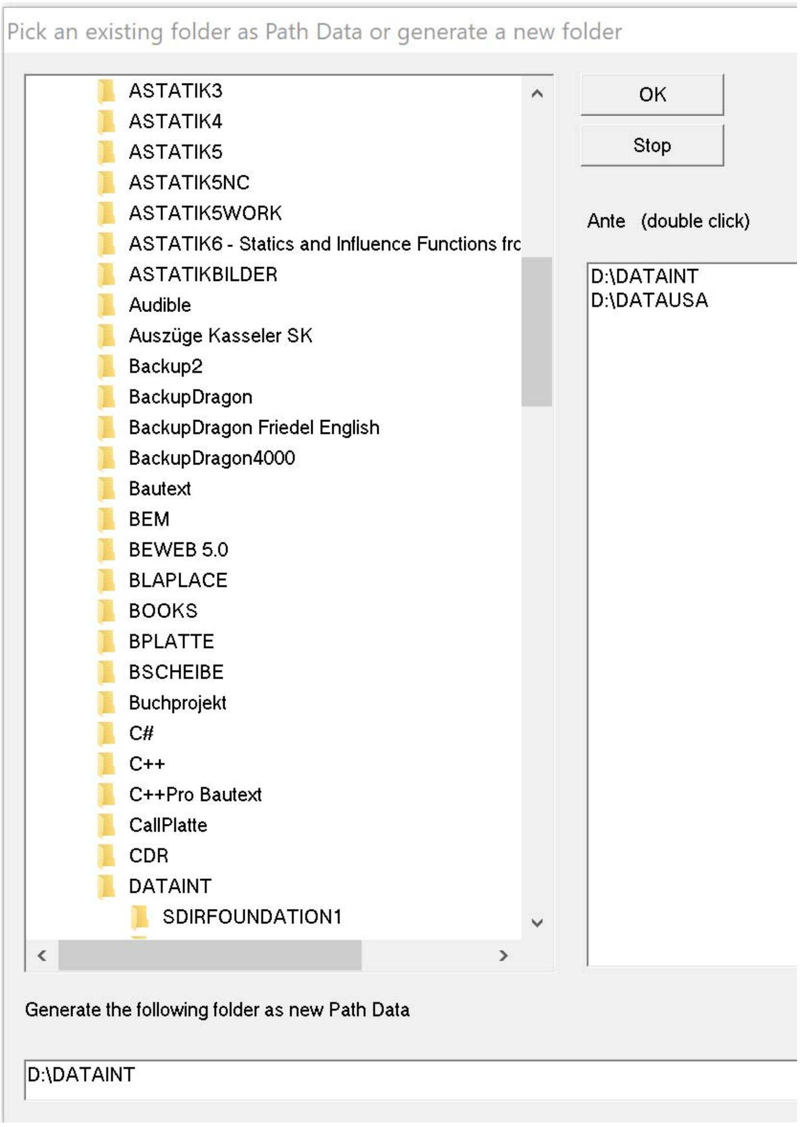


1.4. PATH DATA

The data which belong to position 100 are stored in the folder SDIR100 and the data of a position ABC are stored in the folder SDIRABC, etc. The root directory of these folders is the folder named *path data*. It can lie in any directory on the hard drive or on a network drive as long as the program has write access to it.

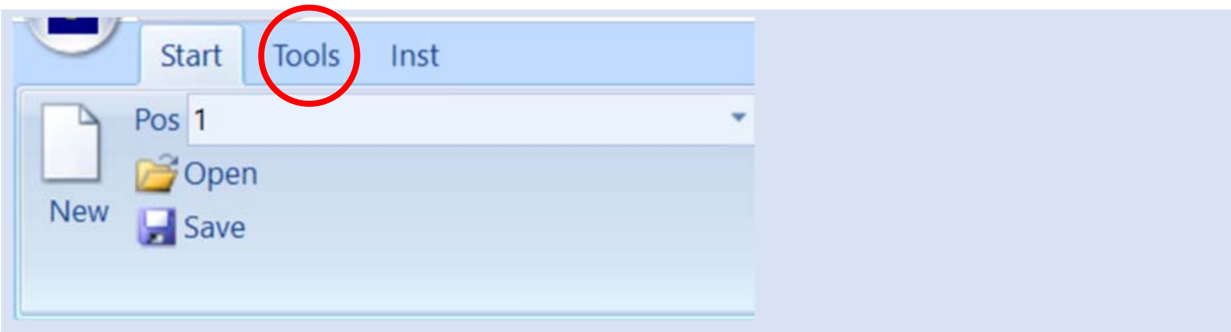
To set or change the path, click on the button

in the main menu.

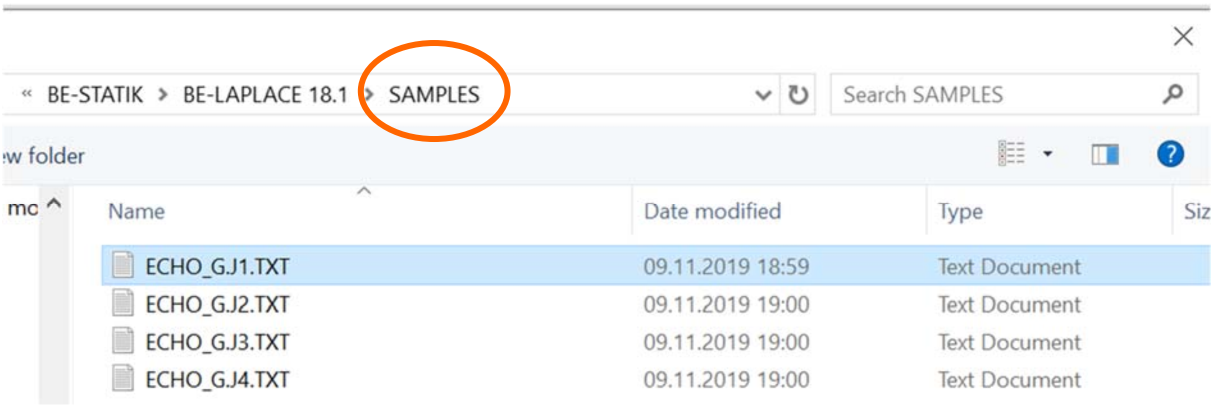


1.5. FIRST STEPS

1. Click on the button Tools



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



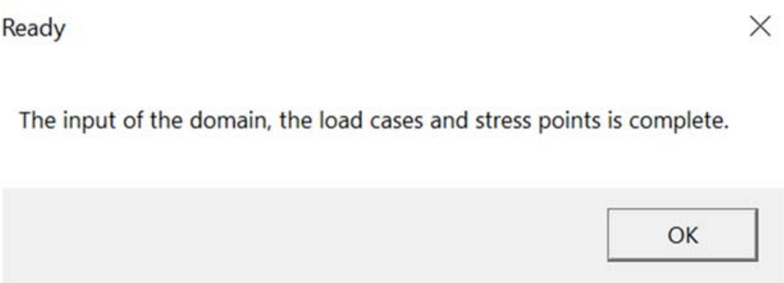
3. Double click on the file ECHO_G.J1.TXT and the position J1 will be installed automatically.

If it fails open the Windows Explorer, go to the folder

› This PC › Volume (C:) › Program Files (x86) › BE-STATIK › BE-LAPLACE 18.1 › SAMPLES

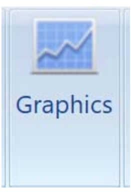
and drag the file ECHO_G.J1.TXT with the mouse onto the open program window.

If all goes well, you read the text message

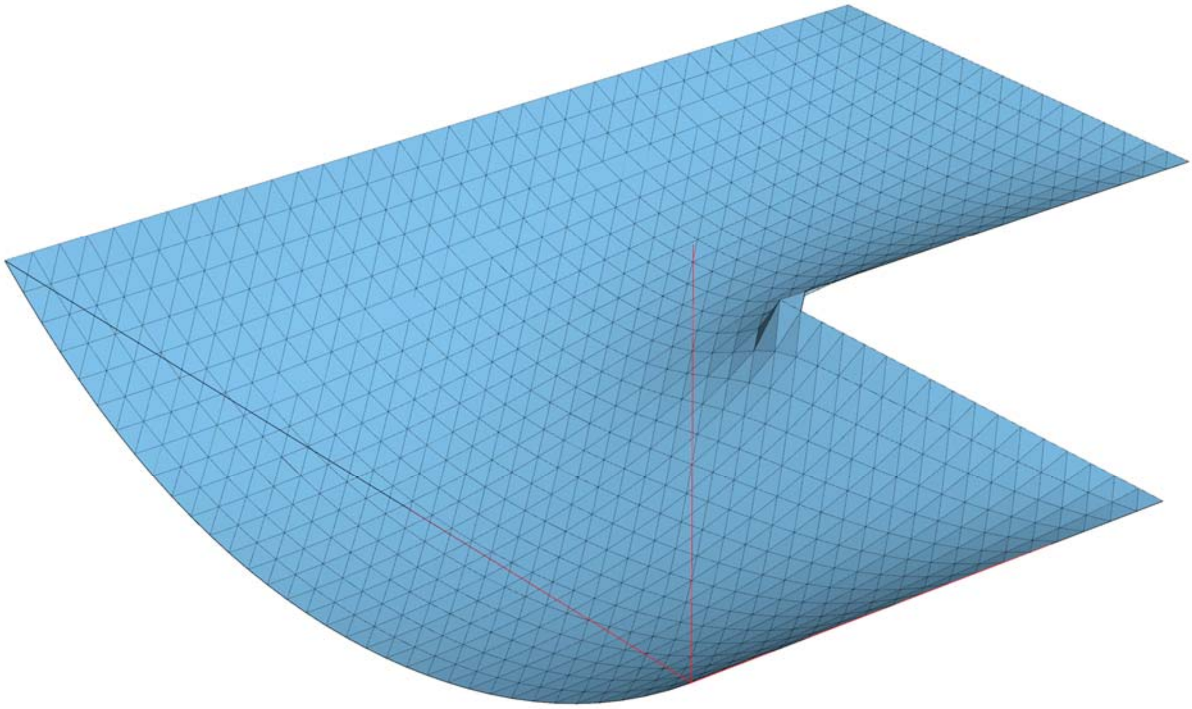


4. Time to start the calculation. In the main menu click on the button

and when the calculation is done click on the button



to display the results on the screen.



Some useful shortcuts in graphics mode:

Mouse wheel scales drawing.

The up and down keys switch between load cases.

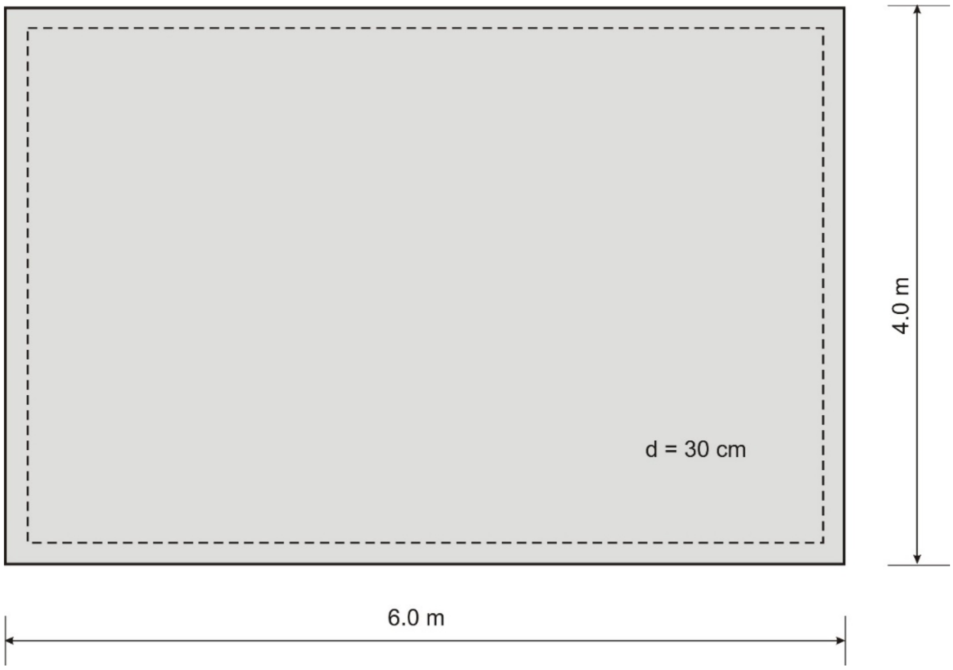
Zoom: click the left mouse button and open a box (window)
Zoom undo: press ESC-key or Right-Mouse-Button

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

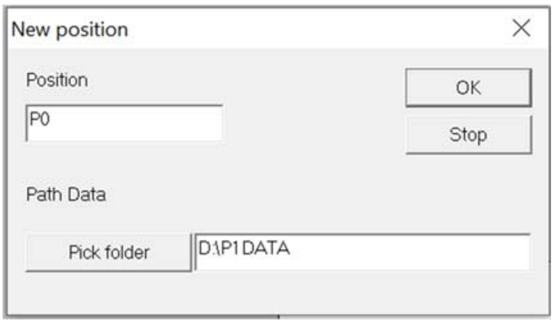
In some windows text (annotations) can be moved around by clicking on the text and pushing the text with the mouse in any direction.

2. INTRODUCTION

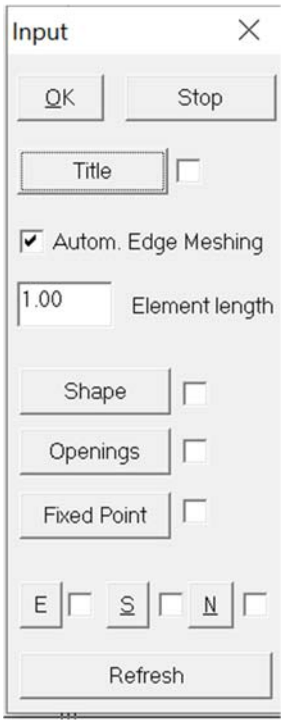
A simple rectangular domain of size 6 x 4 serves as introductory example.



This will be position P0. Click on the button New and enter the text P0.

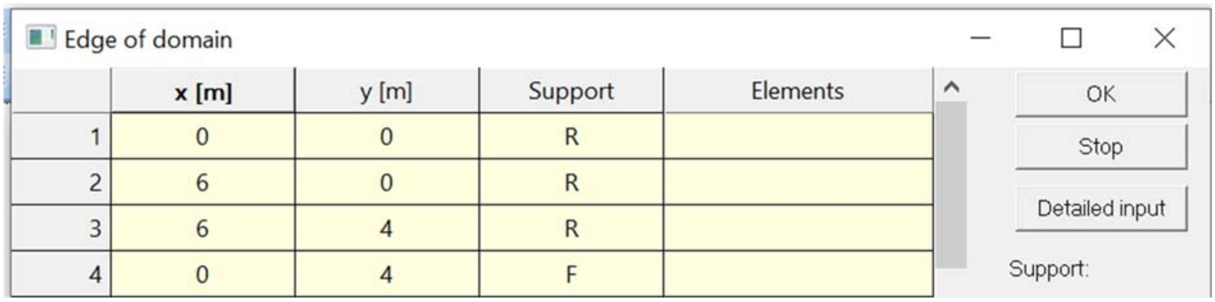


A click on the OK button opens the part of the program where we specify the shape of the domain. We do this in *dialog input mode*, that is we work our way through the following dialog:



If the dialog is not visible activate it by clicking on the entry **Dialog input.** Set the element length to 0.25 m and choose automatic meshing of the edge; alternatively, you could specify the number of elements on each side of an edge separately.

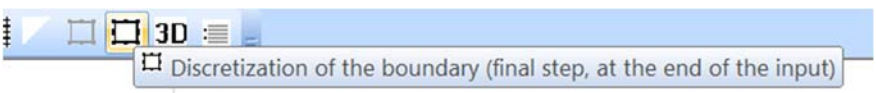
Next click on the button Shape and input the edge of the domain as follows



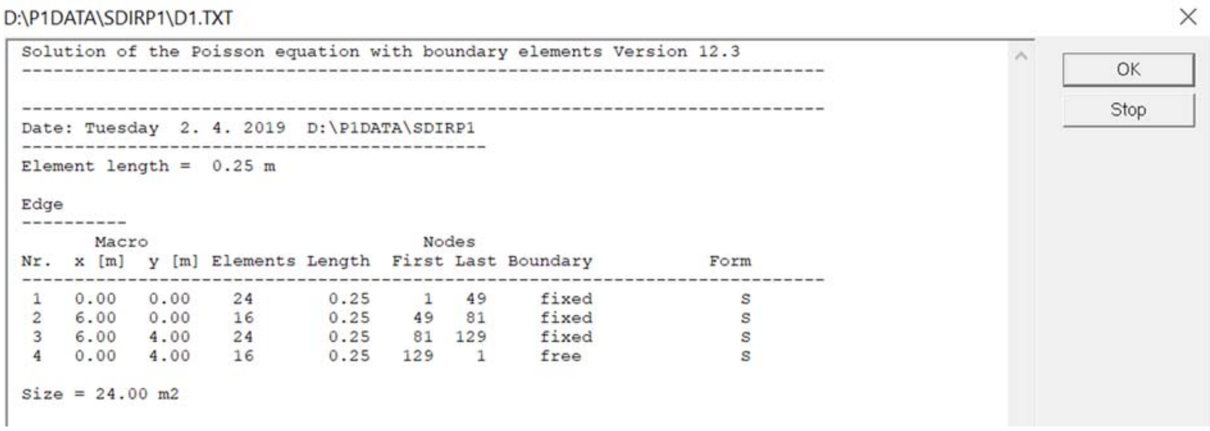
	x [m]	y [m]	Support	Elements
1	0	0	R	
2	6	0	R	
3	6	4	R	
4	0	4	F	

R = rigid, fixed edge and F = free, free edge.

The final step is the click on the icon



to discretize the edge. It is always the last step. A click on the Save button only saves a current snapshot of your input. When the discretization, the subdivision of the edge into elements is done, the input will be displayed in text form

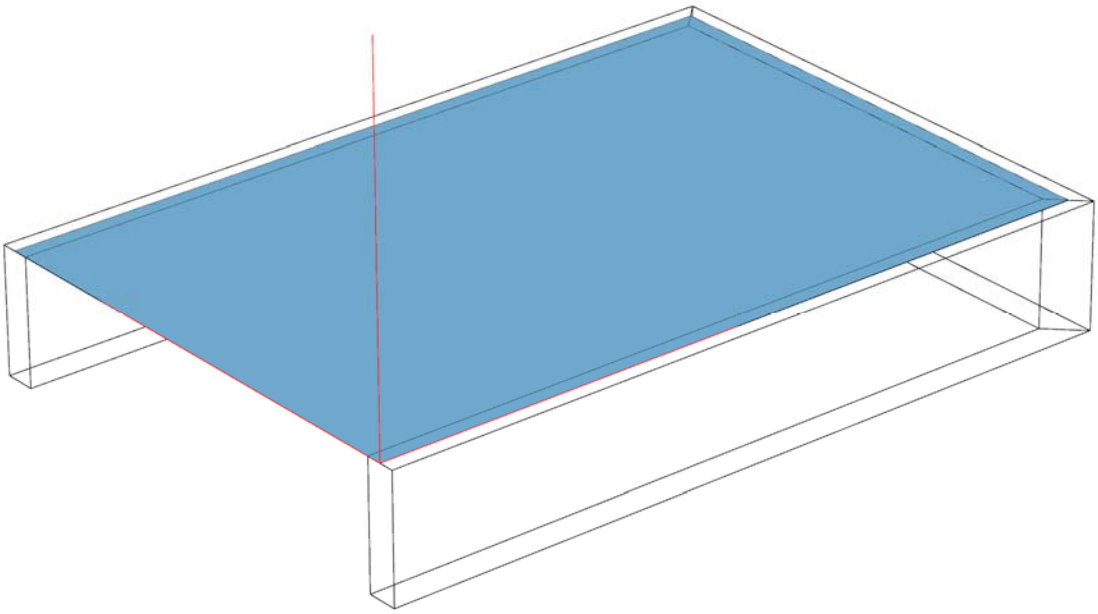


D:\P1DATA\SDIRP1\D1.TXT

```
Solution of the Poisson equation with boundary elements Version 12.3
-------------------------------------------------------------------------------

-------------------------------------------------------------------------------
Date: Tuesday  2. 4. 2019  D:\P1DATA\SDIRP1
-------------------------------------------
Element length =  0.25 m

Edge
----------
       Macro                               Nodes
Nr.  x [m]   y [m] Elements Length  First Last Boundary            Form
-------------------------------------------------------------------------------
 1   0.00    0.00     24        0.25     1    49     fixed              S
 2   6.00    0.00     16        0.25    49    81     fixed              S
 3   6.00    4.00     24        0.25    81   129     fixed              S
 4   0.00    4.00     16        0.25   129     1     free               S

Size = 24.00 m2
```

OK
Stop

and a 3-D figure of the problem domain is displayed on the screen



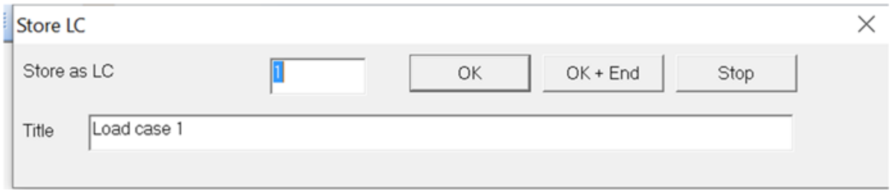
This concludes the input of the domain and we return to the main menu of the program.

2.1. LOAD

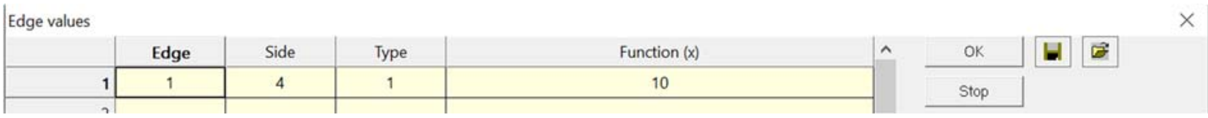
To enter the load (the right side of the differential equation) click on the button LC's

In the first load case we specify a constant load of 5 N/m^2

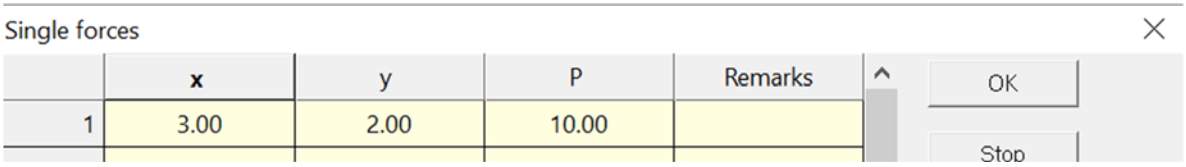
Click on OK and save the load case



Next, click on New to enter the second load case. In this load case we specify the slope on side 4 of the membrane, which is a free edge. At such an edge we can prescribe either the potential u or the slope of u. We prescribe a value of 10 N/m for the slope (Type = 1)



Save this load case, click on New to start the input of load case #3, a point load, a single force



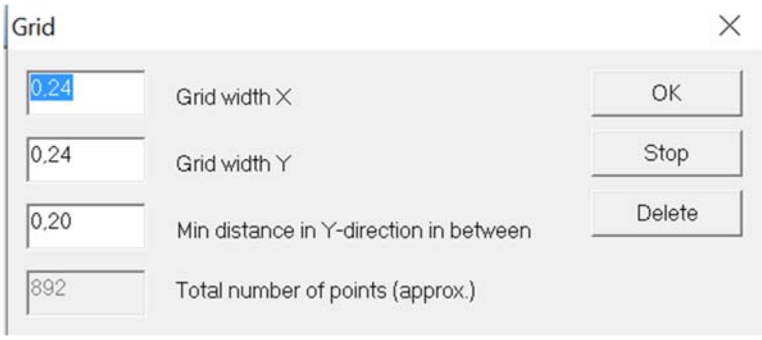
Save this load case and return to the main menu of the program.

2.2. STRESS POINTS

The stress points are so to speak the nodes of the non-existing mesh, the points where the solution is output.

Click on the button Points and then on the (leftmost) red grid symbol

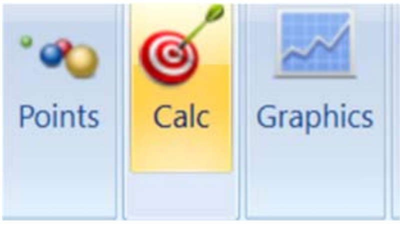
to specify the layout of the mesh



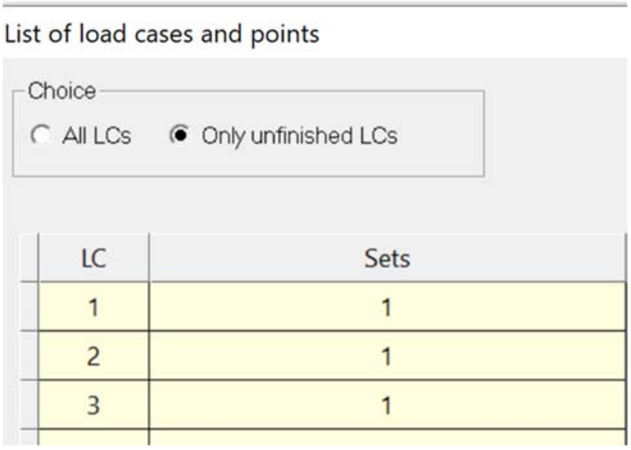
Save and return to the main menu.

2.3. START

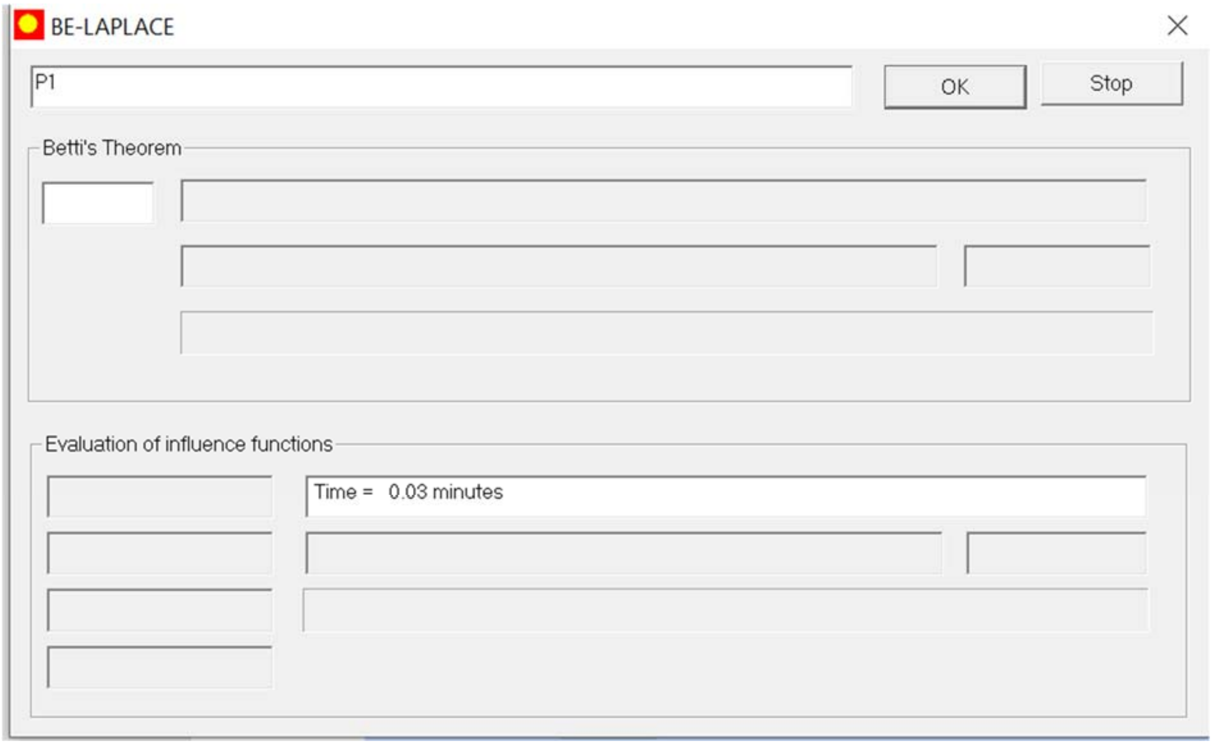
To start the analysis (getting results) click on the Calc button



and in the following dialog click on the button Start of Analysis (not shown in this cutout)



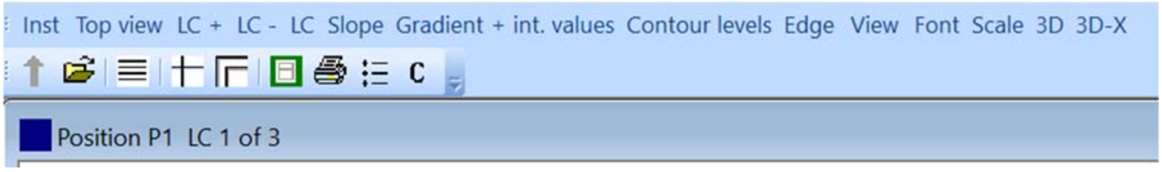
You can follow the action on the screen.

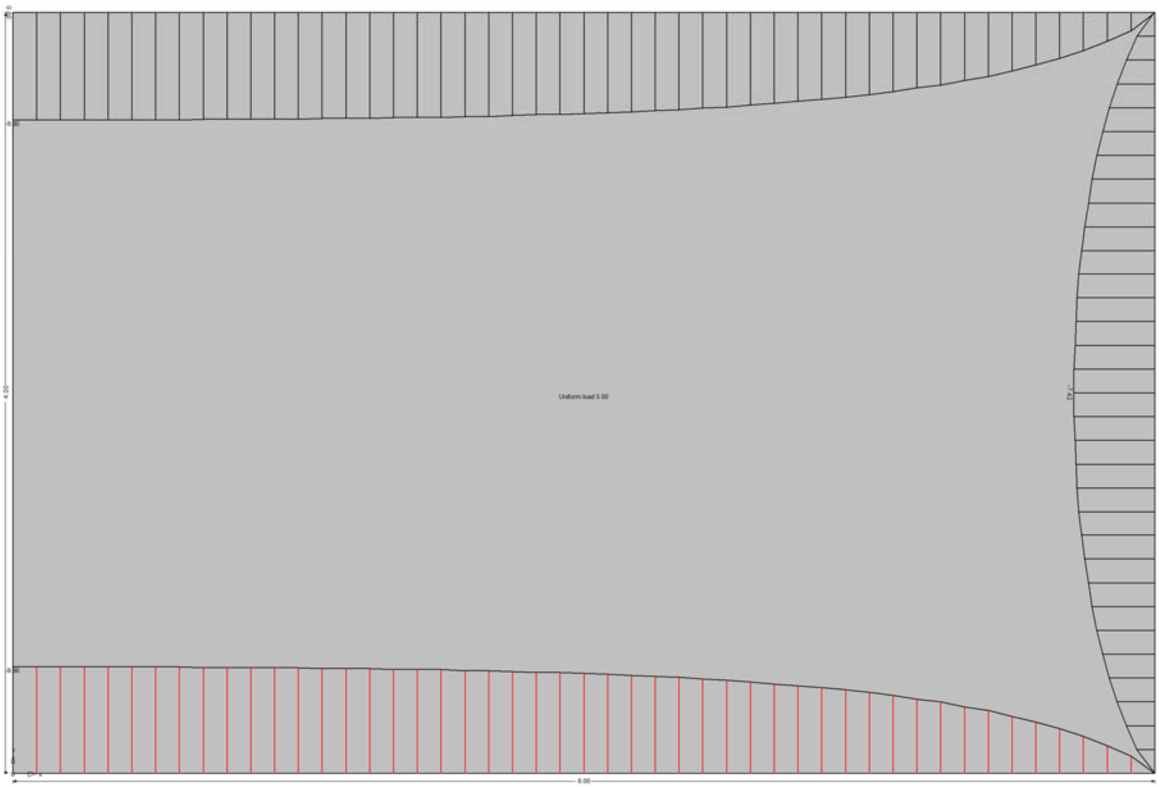


2.4. GRAPHICS

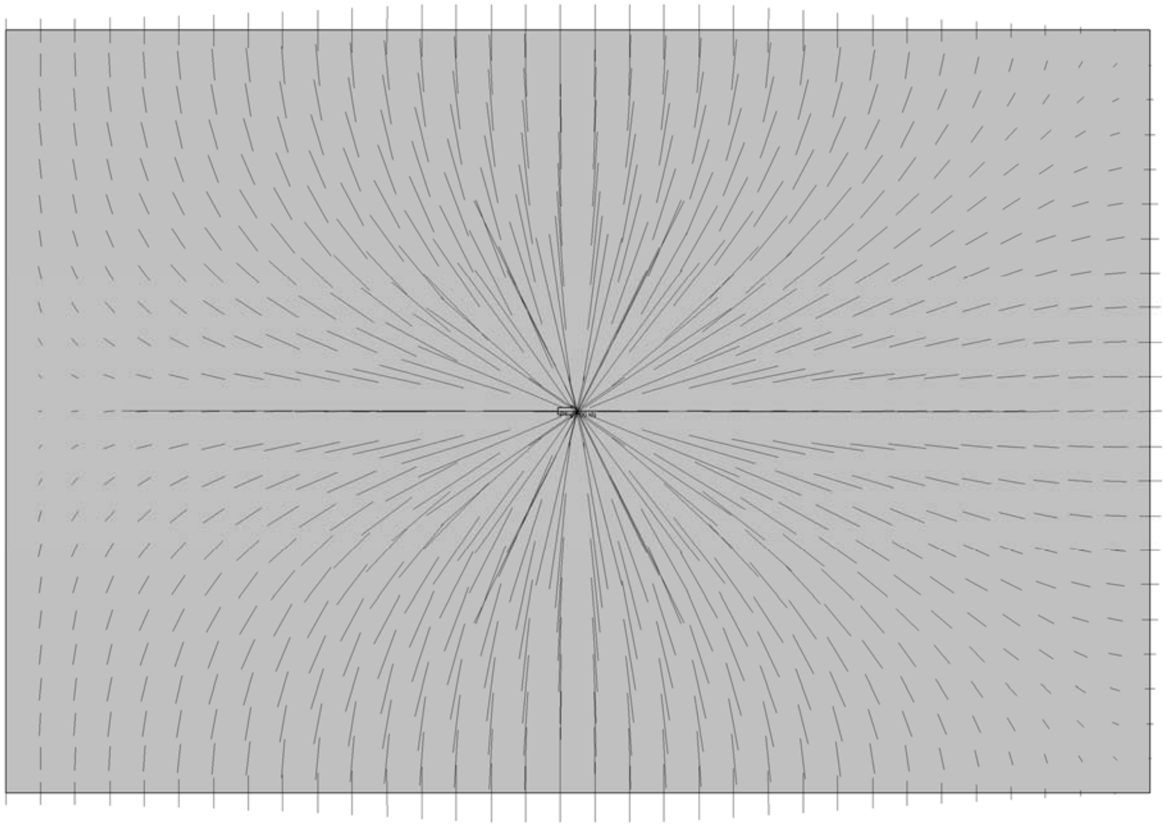
To display the results in the main menu, click on the button Graphics

and pick any of the following options

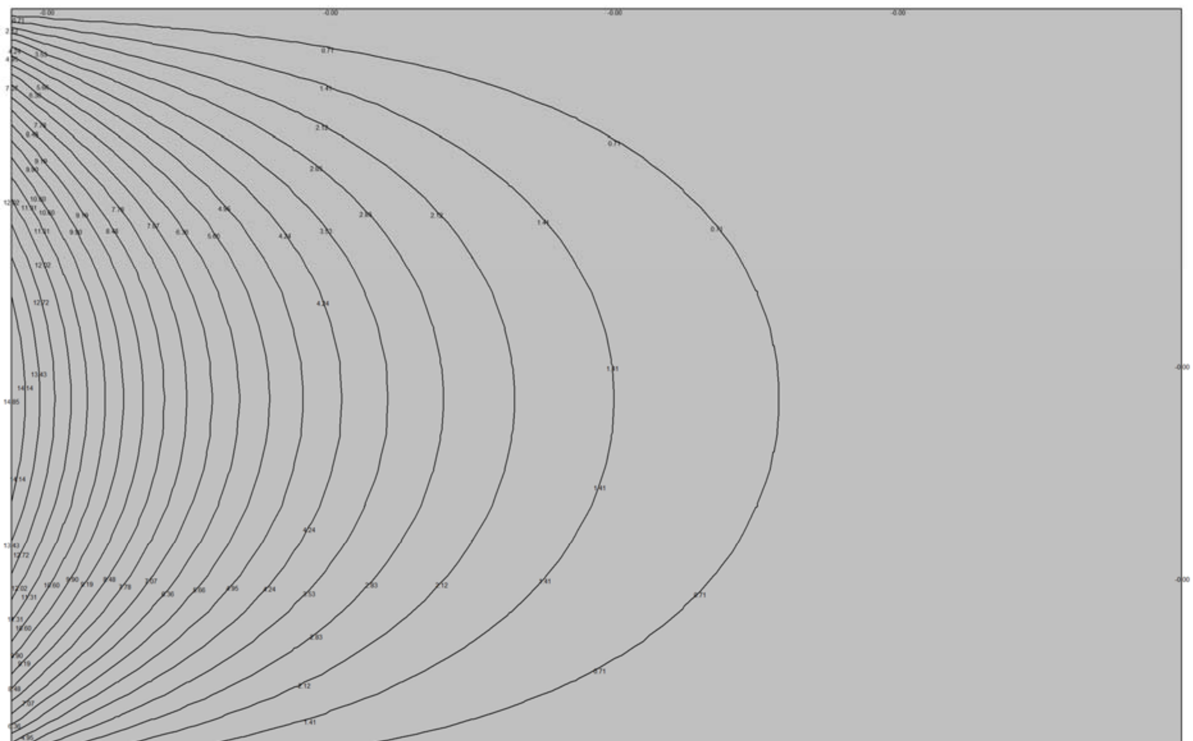




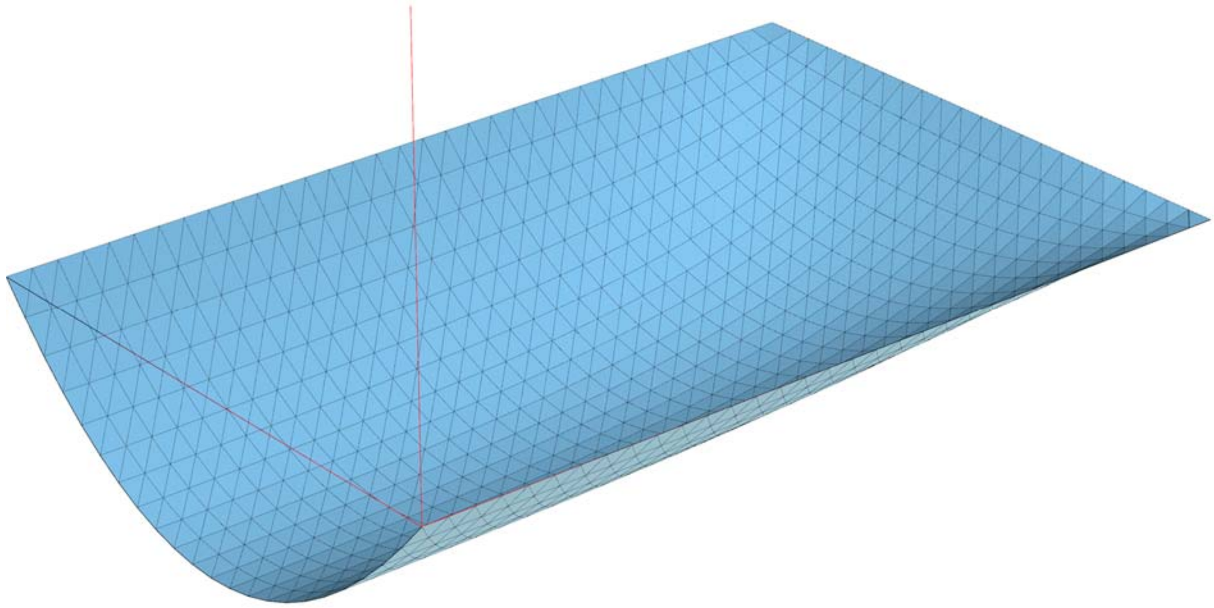
The slope along the edge in LC 1 (uniform load).



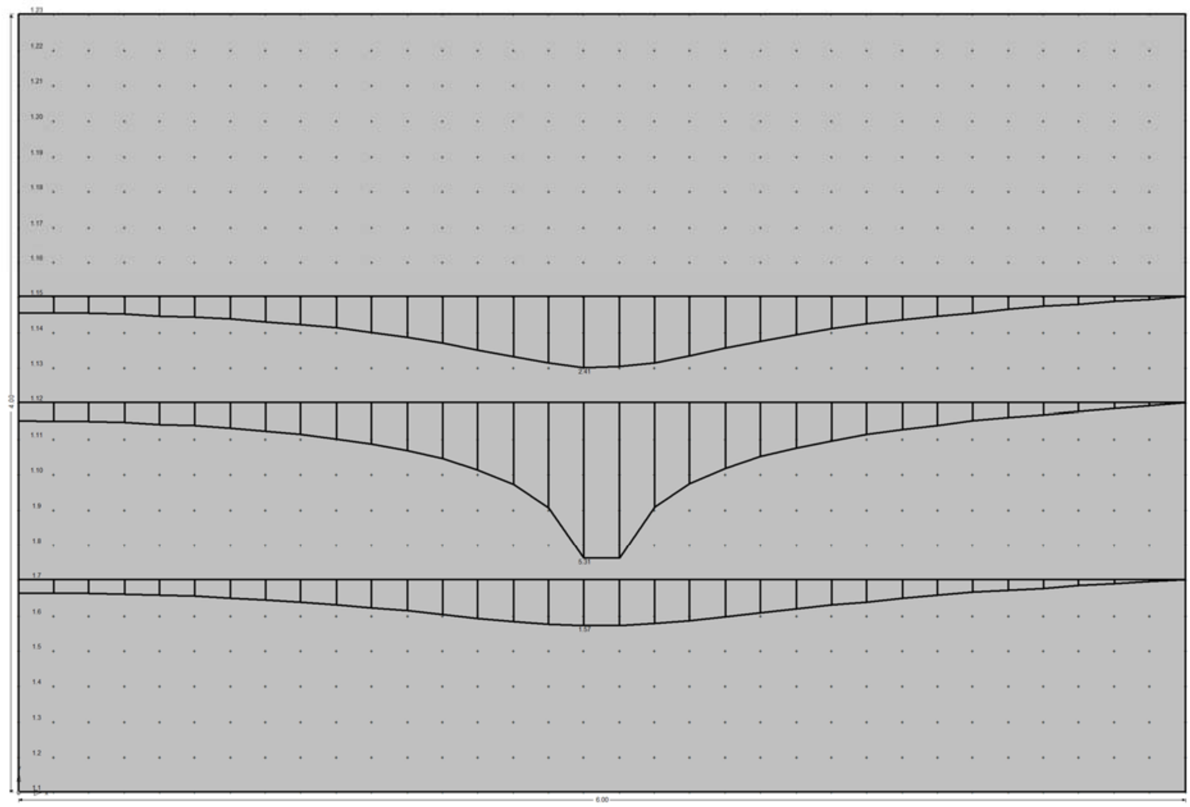
The gradient in LC 3 (single force)



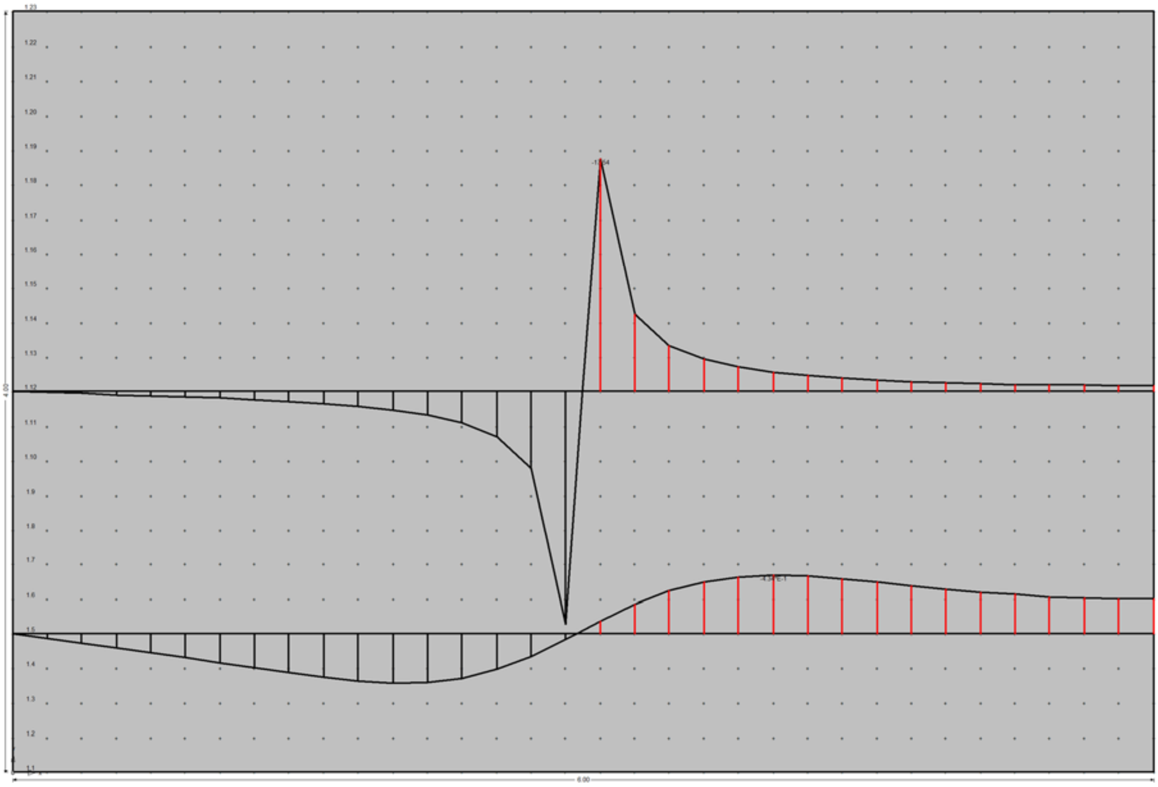
Contour levels in LC 2 (edge load = slope is prescribed on the left side)



3-D View of solution u in LC 1 (uniform load)



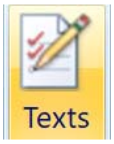
Solution u in LC 3 (Single Force) along specific lines of stress points



Derivative u,x in LC 3 (Single Force) along specific lines

2.5. READING AND PRINTING TEXT FILES

To display or to print the text files with the results of the analysis click in the main menu on the icon



and double-click on the corresponding file.

2.6. ADDITIONAL EXAMPLES

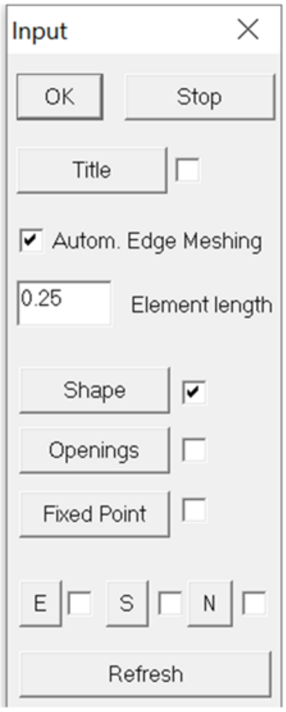
The folder *Samples* contains additional problems for you to try.

3. PROBLEM DOMAIN

The shape of the domain can either be entered with the keyboard (Dialog input mode), or it can be drawn with the mouse (graphical input mode). To start the input, click on the button Dimensions in the main 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

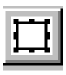


click on the entry Dialog input in the menu bar.

In this mode, you use various dialogs to detail the shape of the domain. It is best to start at the top of the main dialog and to then work your way down through the different dialogs.

If you mix graphical input mode with dialog input mode you must update the entries in the dialogs at the end of the graphical input mode by clicking on the button Refresh.

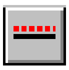
When the input is done, click on the icon



to discretize the edge, to generate the boundary elements. **This is the final and important step.**

3.2. BOUNDARY CONDITIONS

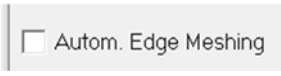
The sides of an edge can either be specified as F = free or R = rigid, that is fixed. In graphical input mode, you can specify the boundary conditions of the single sides by clicking on the icon



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

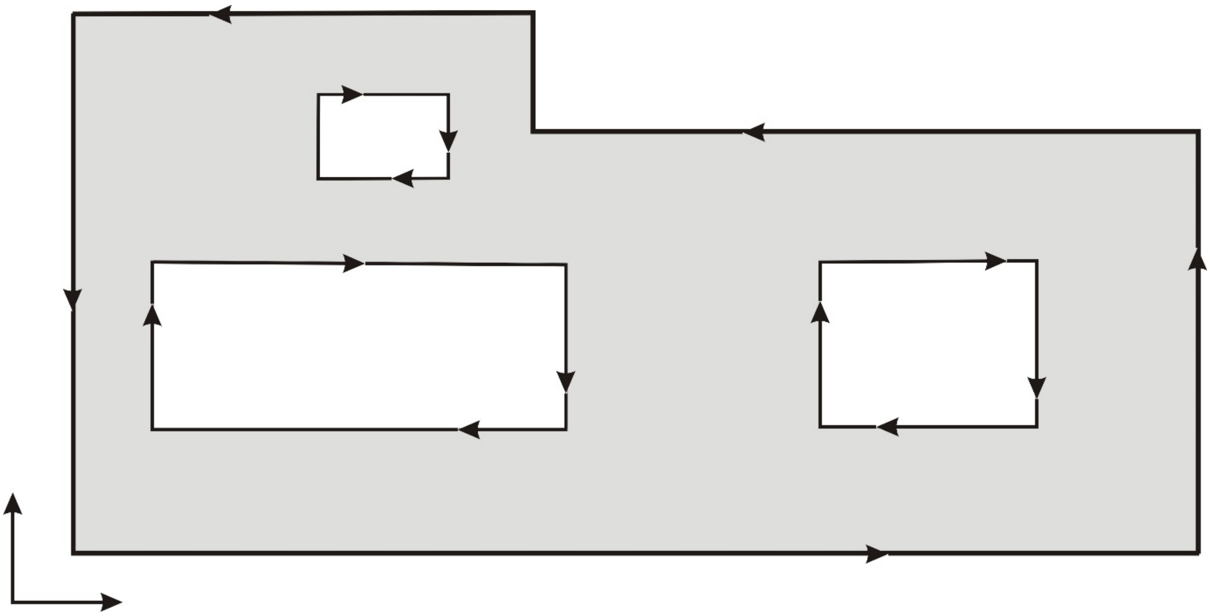
3.3. BOUNDARY ELEMENTS

The program uses quadratic elements. The meshing of the edge is normally done by the program according to the standard element length set by the user. To opt out of this deactivate the button



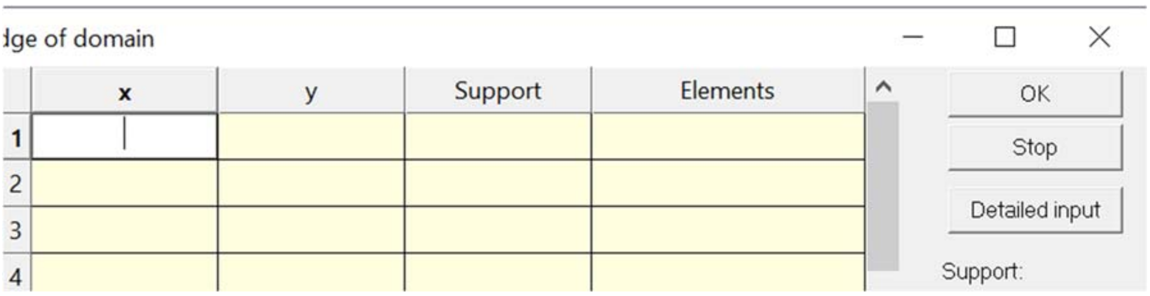
3.4. OUTER EDGE AND OPENINGS

The sense of rotation on the outer edge is counterclockwise and along any interior edge clockwise. The rule is that the domain lies always to the left.



3.5. CURVED EDGE

In dialog input mode, you can enter a curved edge as follows:



Click on the button Detailed input. This will open a more detailed dialog.

Edge

	x	y	Form	Radius	Support	Elements
1	1	2	C	5	R	
2	5	2	S		R	
3						

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.

3.6. GRAPHICAL INPUT MODE

In this mode you click on the icon

to draw a polygon with the mouse. **To close a polygon**, to connect the last point with the first point press the C-button. If the polygon is not closed, you see an error message when the discretization is called.

Boundary conditions can be specified by clicking on the icon

To enter an opening draw the edge of the opening like a polygon and then click on the icon

and then on the center of the opening.

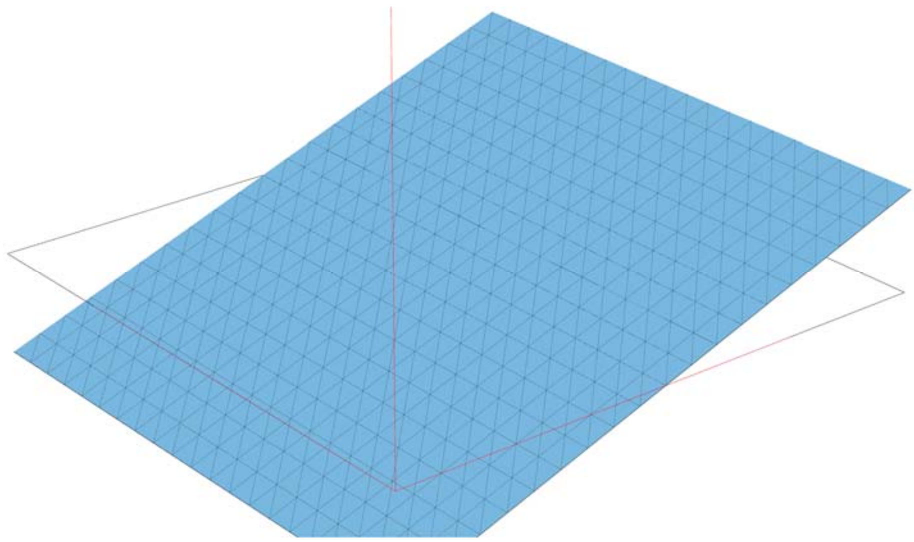
4. ENGINEERING MODE & MATH MODE

As engineers we like to think of the problem domain as a prestressed membrane which along some part of the edge is held fixed while other parts can move freely.

According to the engineers understanding you can only prescribe non-homogeneous boundary data along the free edges but not along the fixed edges, where the only allowed boundary condition, u = 0, is already set. So, a rigid or fixed edge is a Dirichlet boundary with u set to zero.

A mathematician instead wants to have a free choice of what to prescribe where. This is accomplished by specifying free as the boundary condition on all sides. Then Dirichlet or Neumann boundary conditions are possible. The boundary value specified, the input, either a potential (u) or the slope, is considered an 'edge load'.

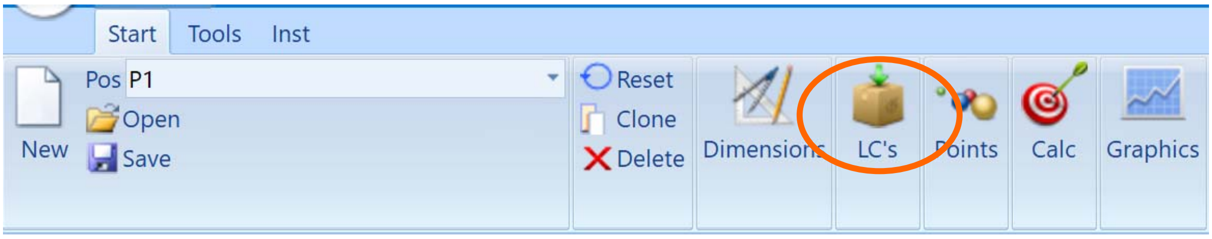
But note that pure Neumann problems (if the slope is specified on all sides) require special care. First, it must be guaranteed that the data satisfy the equilibrium condition, the integral of the slope must be zero, and, secondly, you must provide a fixed point to make the solution of the problem unique – otherwise the system matrix will be singular.



Square domain, pure Neumann problem, slope = +/- 1 along the sides parallel to the y-axis and zero on the other two sides. The domain rotates about a (not visible) fixed point at the center.

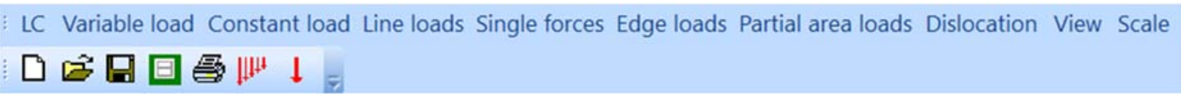
5. LOAD

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

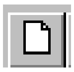


For an engineer the right-hand side of a differential equation is the load, so we use this terminology here

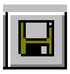
- **Variable load**
- **Constant load**
- **Partial area loads**
- **Line loads**
- **Single forces**
- **Edge forces, edge displacements**
- **Dislocations (influence surfaces for u,x or u,y)**



To enter a new load case, click on the icon



To save a load case, click on the icon



You can also import or copy load cases from other positions

5.1. EDGE FORCES AND DISPLACEMENTS

We imagine the problem domain to be a membrane prestressed by a constant force H = 1 and carrying some load. The sides of the membrane are either fixed or free to move. The fixed sides provide the necessary support. Edge forces, if any, are typically applied along the sides of the membrane which are not fixed, which are not restrained. Edge forces along a fixed edge (u = 0) make no sense and will be ignored.

This is the engineering picture.

A mathematician thinks in terms of boundary conditions for the solution u or its slope du/dn (the normal derivative). To remove any restraints on what you can prescribe on which part of the edge choose free as the default boundary condition on all sides when you input the domain.

To specify the actual boundary values, enter these as 'edge loads'

Edge values

	Edge	Side	Type	Function (x)
1	1	2	1	[sin([x])]
2		3	2	3.1416
3		4	2	1.0
4				
5				
6				

OK Stop

Edge 1 = outer edge
Edge 2 = edge of opening 1, etc.

Type:

1 = slope of solution

2 = solution u

Type 1 corresponds to Neumann and type 2 to Dirichlet. If along a free side no 'edge loads' of type 1 or 2 are prescribed then the boundary condition is set to du/dn = 0, zero slope.

The boundary values (function (x)) can either be entered as text or read in from txt-files.
To enter an edge load as a function f(x) where x = 0 is the first point of the side, follow these rules

f(x) = c
input: c

f(x) = 1 + 3 * x + 4 * x^2 + 5 * x^3
input: [1] + [3]*[x] + [4]*[x^2] + [5]*[x^3]

f(x) = 3 * x + 1.2 * x^2 + 3 * sin(x + 4)
input: [3]*[x] + [1.2]*[x^2] + [3] * [sin([x]+[4])]

f(x) = 2 * (sin(x))^2
input: [2] * [sin([x])]^2

f(x) = 2 * ln(x)
input: 2 * [ln([x])]

f(x) = 1 * e(x)
input: [1] * [e([x])]

Or start MATLAB and generate a *.txt file, say ABC.TXT, which contains the nodal values

0.000 4.567 8.910 9.112 … (blank-separated)

of your input. (m elements on a side have 2*m+1 nodes since quadratic elements are used).

Edge values

	Edge	Side	Type	Function (x)
1	1	6	2	ABC.TXT
2	1	5	2	ABC.TXT
3	1	1	2	DEF.TXT
4	1	2	2	
5	1	3	2	
6	1	4	2	DEF.TXT

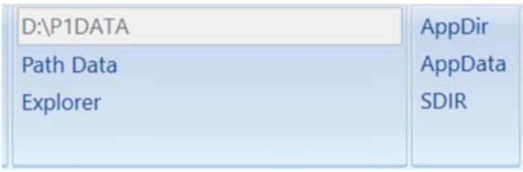
If no file is specified:

At sides with boundary condition 'rigid', u = 0 at all nodes
At sides with boundary condition 'free', the slope = 0 at all nodes

MATLAB-Example f(x) = x*(x-2)*(x-4)*(x-6)*(x-8)

```
elements = 32;  % the side consists of 32 quadratic elements
length = 8;     % length of the side
deltax = length/(elements)/2;   % distance between nodes

x = 0:deltax:length

f = x.*(x-2).*(x-4).*(x-6).*(x-8)

save('ABC.TXT','f','-ascii')
```

Store the file in the subdirectory SDIR… which belongs to the position. You can open this subdirectory when you click on the button SDIR



in the main menu of the program.

In the program folder lies a more user friendly version GenEdgeInputMatlab.mlx of this code.

5.2. VARIABLE LOAD

The program replaces a non-uniform load by a series of point loads placed at the stress points of a mesh that extends over the whole domain.

Step #1 Generate a uniform mesh of stress points (max = 3000 points) and store it as set 1 (must be set 1)

Step #2 Start MATLAB, edit the script GenLoadMatlab.mlx and run the script

Step #3 Start the load case input (LC's), choose Variable load, and enter the name of the file which MATLAB has generated. Here this file is called MATLABOUT.

Save the load case and start the calculation (Calc).

The size of the point load at a stress point x, y corresponds to the value of the load function f(x,y) (specified in MATLAB) at this point. If deltax and deltay is the cell size of the grid you should multiply the load function in MATLAB with deltax * deltay. The point loads would then correspond to a one-point quadrature of these small 'block loads'.

The file GenLoadMatlab.mlx lies in the program directory.

The MATLAB code GenLoadMatlab.mlx

```
clear
```

Edit the following 3 strings

```
sRoot = 'D:\P1DATA';  % <------- EDIT sRoot is the DataPath, the root directory of your folders SDIR...
sPos = 'J7A';         % <------- EDIT sPos is the name of the position
sOutputfile = 'MATLABOUT'; % <------- EDIT  Matlab outputs the data to this file in the folder SDIR...
```

Reading the number of stress points (points of the mesh)

```
sFile1 = strcat(sRoot,'\SDIR',sPos,'\nPoints');
fileID = fopen(sFile1);  % number of points
n = fread(fileID,[1 1],'int16');
fclose(fileID);
```

Reading the coordinates x, y of the stress points

```
sFile2 = strcat(sRoot,'\SDIR',sPos,'\SPMATLAB');
fileID = fopen(sFile2);  % coordinates of the stress points
A = fread(fileID,[2 n],'float'); % A (2xn) row 1 = x, row 2 = y of n stress points
fclose(fileID);
```

Function, edit the line f(i) = ...

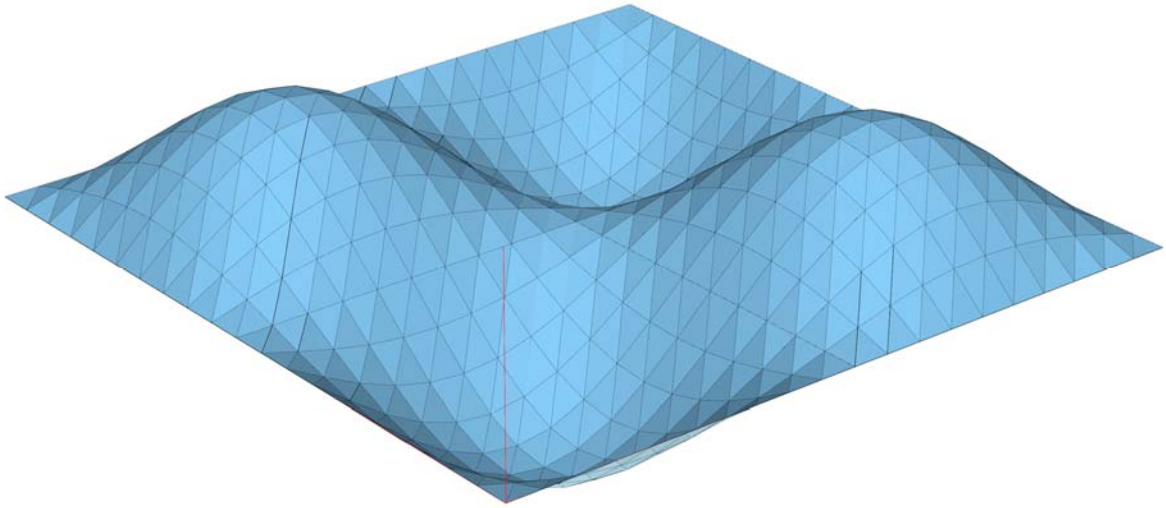
```
f = zeros(1,n);  % in this row vector of length n will be stored the function values

for i=1:1:n
  x = A(1,i);
  y = A(2,i);
  f(i) = sin(pi * x/2) *  sin(pi *y/1) * 5; % <----- EDIT, sine waves in x- and y-direction, amp = 5
end
```

Matlab generates a matrix B (3xn), 3 rows, n columns, first row = x-coord. second row = y-coord. third row = function value at (x,y)

```
B = zeros(3,n);

for i=1:1:n
  B(1,i) = A(1,i);    % x
  B(2,i) = A(2,i);    % y
  B(3,i) = f(i);      % force
end
```

output

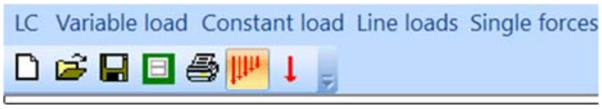
```
sFile3 = strcat(sRoot,'\SDIR',sPos,'\',sOutputfile);
fileID = fopen(sFile3,'w');
fwrite(fileID,B,'float');
fclose(fileID);
```



Solution of $-\Delta u = sin(\pi * x/5) * sin(\pi * y/5) * 5$ in a square of size 10 x 10, fixed edges

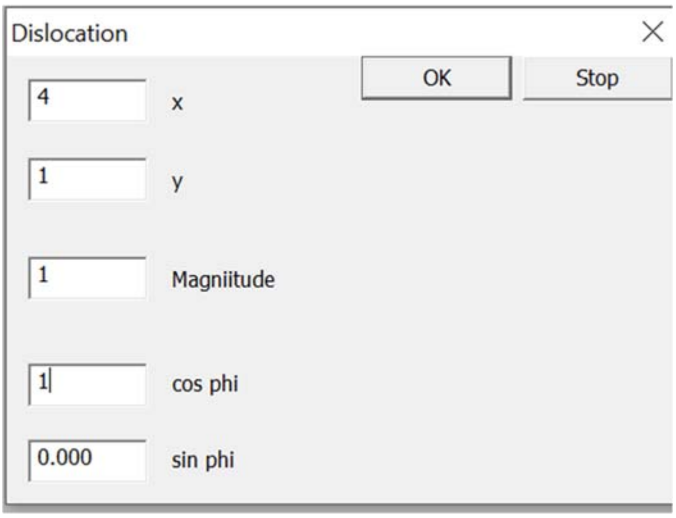
5.3. LINE LOADS AND POINT LOADS

Line loads and single forces can also be entered with the mouse by clicking on the corresponding icons



5.4. DISLOCATIONS

Dislocations

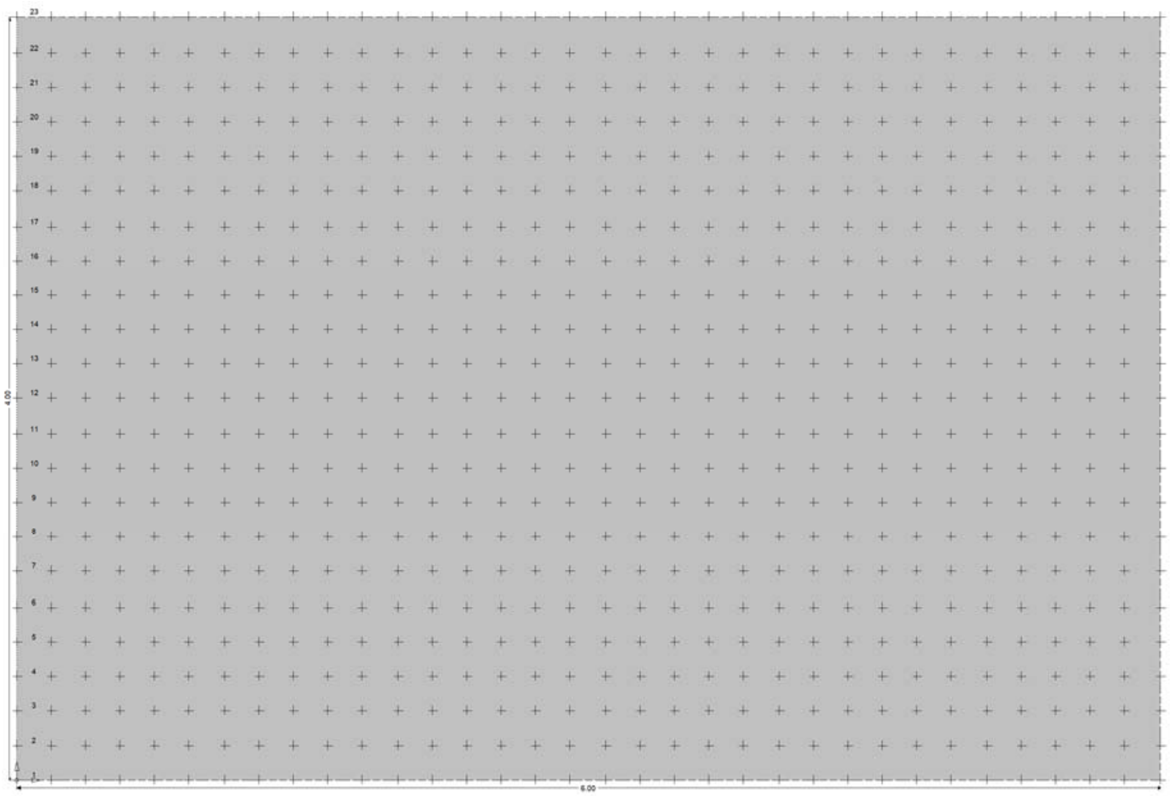


here in 'horizontal' direction (cos phi = 1) generate influence function for the slope in the direction specified by the vector (cos phi, sin phi)/||length of vector|| (make it a unit vector)

6. STRESS POINTS

6.1. INPUT OF STRESS POINTS

The stress points are the nodes, the Gauss points, so to speak, in a BE-program where the solution u and the gradient, that is u,x and u,y, are printed. Because there is no mesh in a BE-Program these points must be generated separately. Since the accuracy does not depend on the mesh size the grids or meshes can have any shape and any size.



The stress points can form a complete mesh or a partial mesh, they can form straight lines, or they can be isolated single points with no apparent recognizable pattern in their arrangement.

Stress points are stored in different sets.

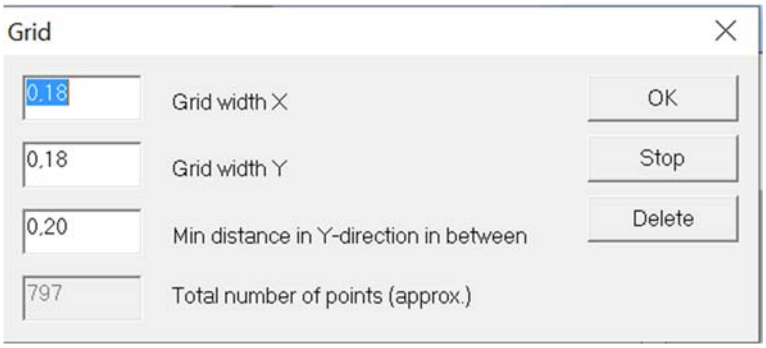
#1: The first set is usually a regular mesh of stress points which covers the whole domain.

#2: Set number two may consist of a set of single lines of stress points in specific directions.

#3: Set number three may consist of single points

You can have up to nine different sets of points.

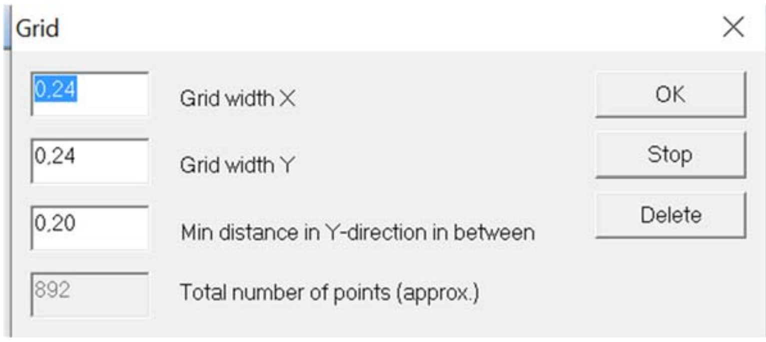
To input a mesh, click on the red grid icon. The program will try to align the mesh lines near openings in such a way that the lines will touch the lower or upper edge of an opening. If the edges of two openings – sitting side by side - are almost on the same level this would force the mesh to have two 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. Each point is independent in the sense that the solution at each point is calculated by influence functions which (mainly) live on the boundary and not by interpolating values in between the stress points.

6.2. LET THE PROGRAM DO THE WORK

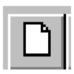
The program will generate the default sets of stress points automatically for you - if you want.



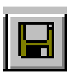
But you can do it also yourself, of course.

6.3. NEW SET

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

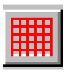


and to store the set – after it has been generated - click on



6.4. GRID

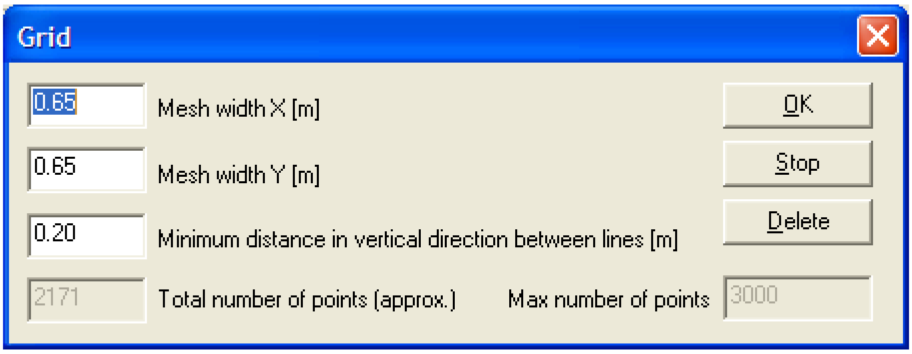
By clicking on the icon



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

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

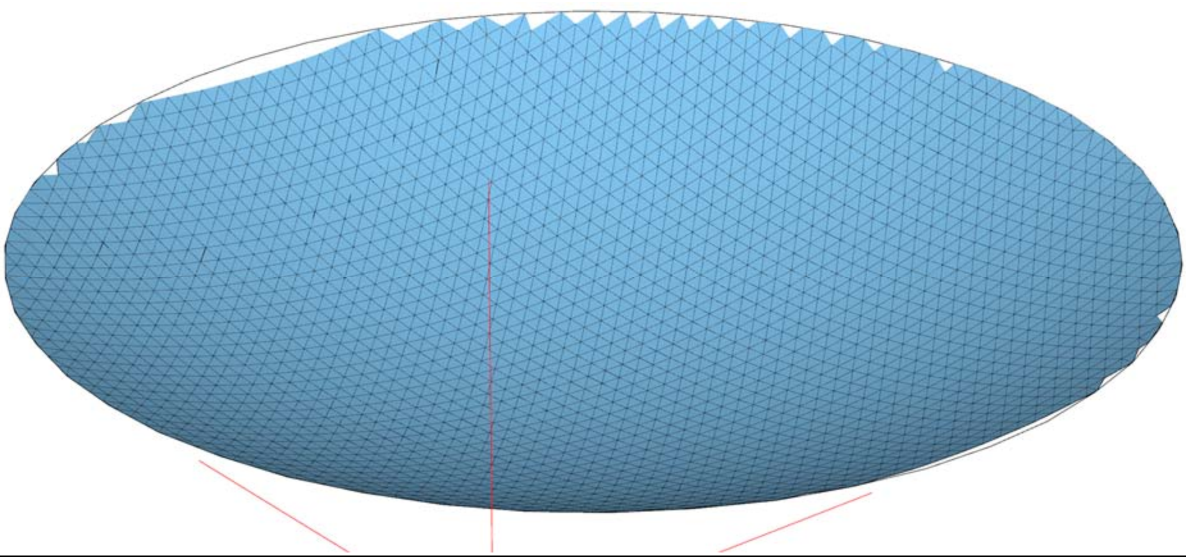
The grid (mesh) is determined by the mesh width in horizontal and vertical direction. The grid actually is not a mesh but consists of a set of horizontal lines on which the points are distributed in (approximately) equal distance.



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

6.5. CURVED EDGES

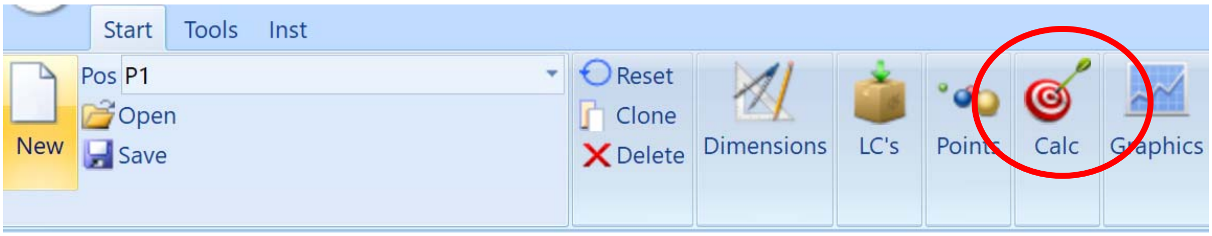
At curved edges the grid cells will not extend exactly up to the edge, but small gaps may be visible. This is not a problem of the solution but rather of the grid which consists of small cells, (four stress points form a cell). Near a curved boundary this formation, sometimes, is untenable.



6.6. STRESS POINTS VERY CLOSE TO AN EDGE

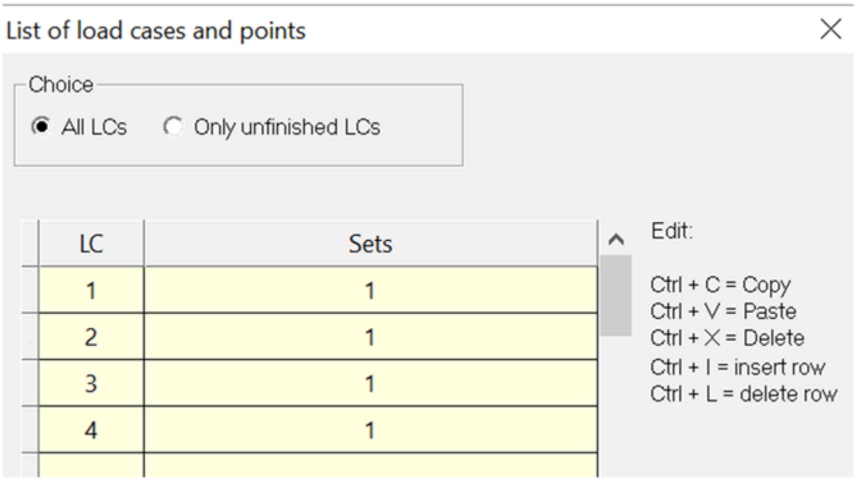
The program uses quadratic elements and a 10-point quadrature. Normally, with straight vertical and horizontal edges the distance of a stress point to the edge is sufficiently large but near slanted edges the stress points may come too close to the boundary and the accuracy will suffer. In this situation you should refine the mesh size on this part of the edge.

7. START OF ANALYSIS



After the input of the domain, the load and the stress points you start the analysis by clicking on the button **Calc** in the main menu.

7.1. START



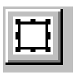
Here are listed the load cases the program will analyze and the sets of stress points at which stresses and displacements are calculated. This table can be edited.

7.2. MODIFICATIONS

After a successful run of the program you can, of course, input additional load cases or generate new sets of stress points.

- **Modifications of the domain itself**

After any modifications of the domain, the layout of the mesh or a change in the support conditions you must call the discretization of the domain anew by clicking on the icon



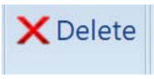
and then restart the calculation. Prior to this it is advisable to reset the problem by clicking on the icon

to delete the previous text-files.

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

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

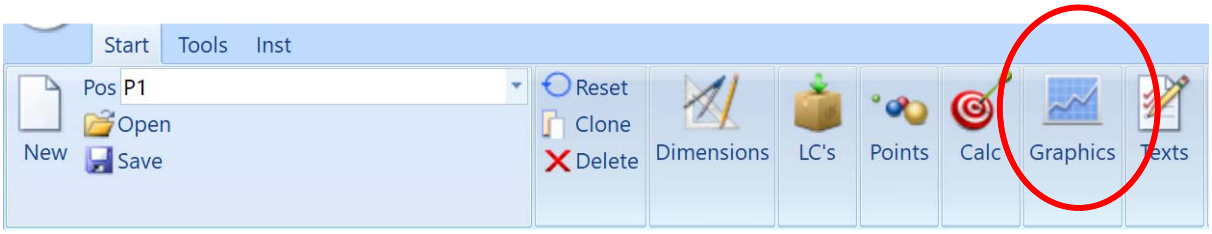
While resetting a position or duplicating a position leaves the position intact a click on the icon



will delete the position from the disk.

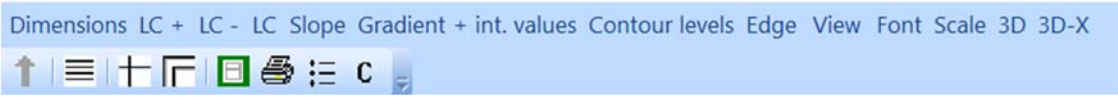
If you want to save your input before you delete any files, click on the icon

8. DISPLAYING RESULTS



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

This will open the program GRAPHICS program



where the results can be displayed

8.1. HANDLING

Changing the scale

By pressing the plus and minus key on the number block, you can scale the drawing on the screen.

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

Mouse wheel scales drawing.

up and down keys = switch between load cases.

Zoom: left mouse button and open a box (window)
Zoom undo: press ESC-key or Right-Mouse-Button

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

Printing

The contents of the screen can be printed, plotted or copied to the clipboard (icon C).

Print preview: green edged printer icon

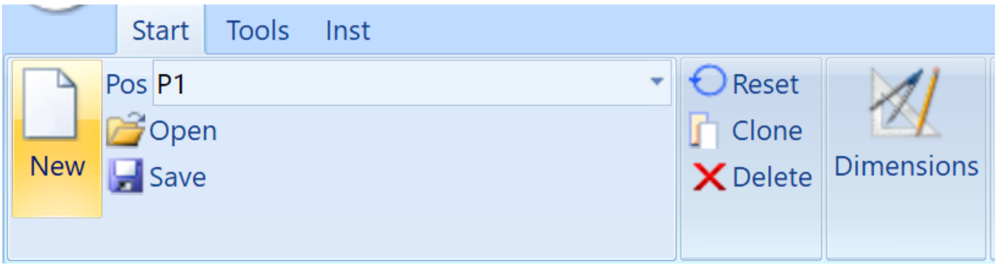
Context menus

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.

9. STORING AND RETRIEVING



9.1. 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.

9.2. STORING ONLY THE INPUT

Otherwise you can only store the input of a position. This will allow you to re-load the position from the archived files.

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

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

The program stores the input in three small text files

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

9.3. RETRIEVING A POSITION

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

ECHO_G.200.TXT

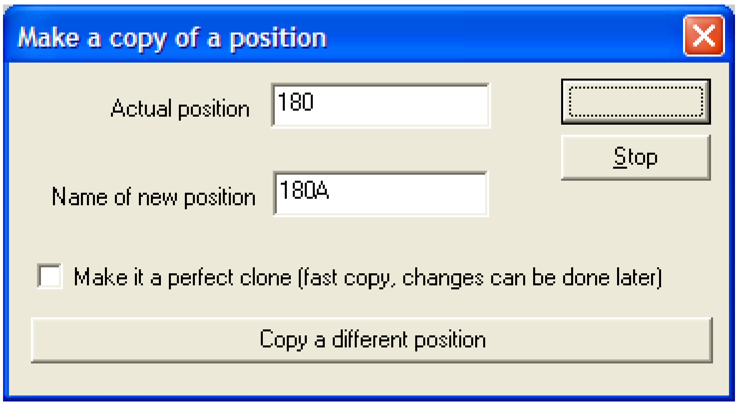
and drag it onto the open program window.

The program will load automatically also the additional files ECHO_L.200.TXT and ECHO_S.200.TXT from the same source.

9.4. DUPLICATING A POSITION

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

and enter a new name for the copy



The following will happen:

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

Next the program loads the shape of the domain from the file ECHO_G.180A.TXT and displays it on the screen so that you can modify the shape or the boundary conditions.

The same happens with the load cases and the stress points. They are copies of the original position, but they can be modified as needed.

9.5. RESETTING A POSITION

A click on the icon

will reset the actual position, will delete the previous results but not the input.