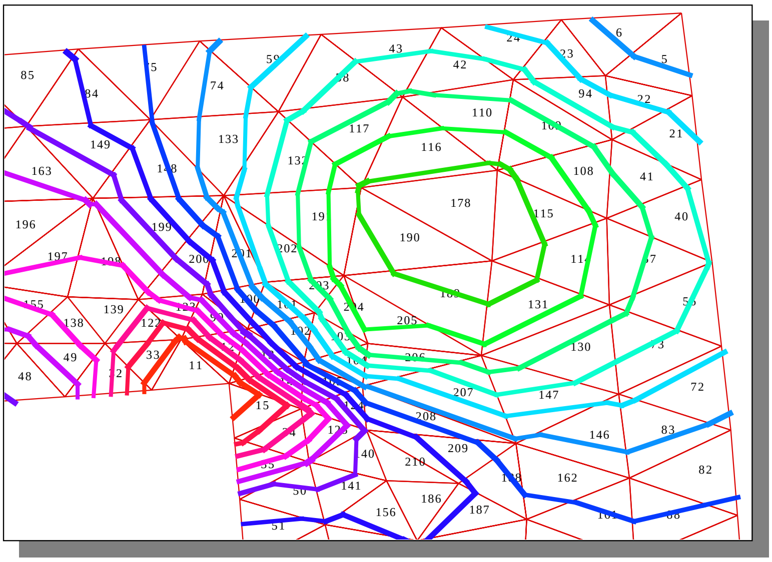
PlastFEM

v2.0

Finite Element Program for Mac OS™

Manual



Utility Apfel Research Kitchen

PlastFEM

Finite Element Program for Mac OS™
v2.0

by Dirk Fröhling

Documentation

S. Kessel

D. Fröhling

Special thanks to Igor Mikolic-Torreira

1st printing

Table of contents

1. Introduction

The finite element method currently is the best known method for the analysis of continuum problems. In the field of mechanics it is used for the solution of static and dynamic, linear and nonlinear elastic or plastic problems. Depending on the particular case, a variety of different theories of solids, structures and dynamics can be applied, and lots of element types can be chosen from for discretization.
The ability to deal with problems that are not analytically solvable – for example geometrically complex systems – is of great importance for the engineering practice. The progress in computer techniques made it possible to analyse realistic structures with sufficient degrees of freedom and – for nonlinear problems – number of calculation steps using simple desktop computers in an acceptable time frame.

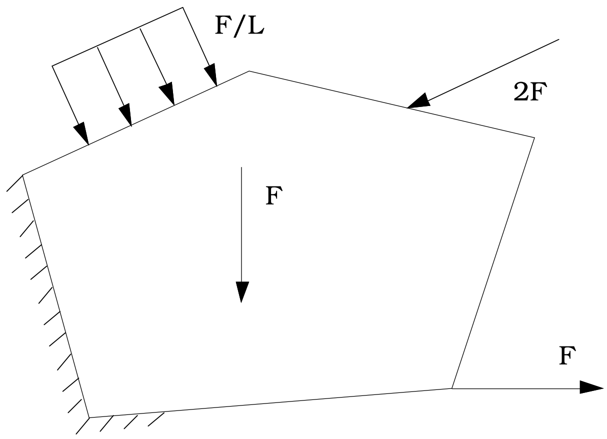


Fig. 1.1: Example of a plane stress problem (thin plate under in-plane loading)

The finite element program presented in this manual, *PlastFEM*, is capable of dealing with two-dimensional, isotropic, elasto-plastic plane stress problems, that is, thin plates loaded in the plane of the plate. The limitation to this relatively small part of the analysis of solids, together with the graphical input/output capabilities of the program, should make it easier to get introduced to the inner workings of the finite element method and the way of using it.
The calculation part, which is based on a linear-elastic program by S. Kessel, is embedded in a graphical user interface with menu and dialog control, which allows interactive input, checking and correction of the problem description, as well as the graphical analysis of the calculation results (displacements and stresses).

PlastFEM has been written in Pascal and C and works on all Mac OS computers with Color QuickDraw and System 7 or higher. There are three versions: *PlastFEM* runs on all these machines, *PlastFEM 881* is for Macintosh computers with a numerical coprocessor, and *PlastFEM PPC* is for PowerPC.

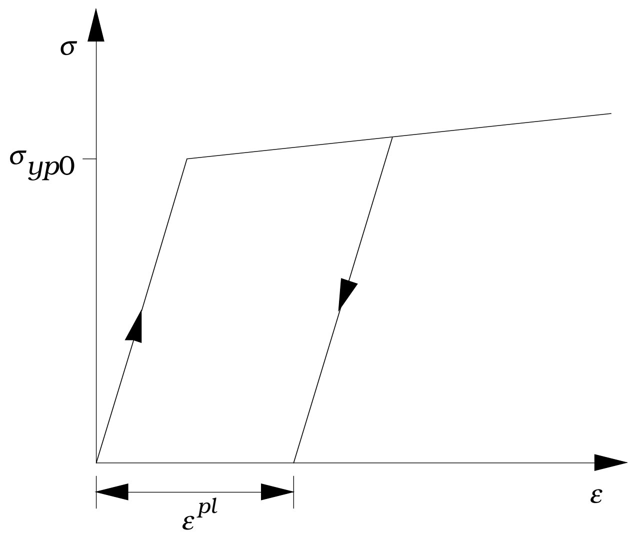


Fig. 1.2: Non-linear stress-strain relation with hardening

The initial stress method, also called modified NEWTON-RAPHSON method, is used for plastic calculations. This technique takes ideal-plastic behaviour of materials, hardening and softening into consideration (Fig. 1.2) and has been developed by Zienkiewicz, Valliapan and King/10/. The elements used for discretization are triangle elements with linear interpolation functions.

This manual is divided into seven chapters. Chapter 2 is a reference guide to the program, chapter 3 discusses some example problems, and chapter 4 gives an introduction to the theory of the finite element method. Chapter 5 deals with the solution method for a system of equations which is used here, and chapter 6 describes the basics of the theory of plasticity. Chapter 7 contains some references to the literature. For a general introduction into the mechanics of solids, which builds the basis of the finite element method, you can look at the textbooks from Benham, Crawford and Armstrong/2/ or Kessel and Fröhling/3/ (shameless plug ;-). A more exhaustive description of the finite element method, including the mathematical basics, is given by Bathe/1/, Meek/4/ or Zienkiewicz/9/.

It isn't necessary to work through chapter 4 to 6 just to use *PlastFEM*. This part of the manual is meant for people who want to learn about the finite element method by using the program and the examples. However, it should be mentioned that knowledge about the theory can greatly help you to judge the possibilities and limitations of this method.

2. Reference

2.1 Introduction

The program uses the Macintosh toolbox for providing the user interface, so the basic functions work similar to other Mac OS programs.
PlastFEM needs a place to store data files in order to do calculations. We recommend using a dedicated folder for this purpose, since many files will get created and these files are useful for *PlastFEM* only. Without a data path to such a folder, *PlastFEM* can only show pictures and edit text.
The graphics created can be copied to the clipboard for use in other programs or saved as PICT files, which are compatible with most drawing programs.
You can start *PlastFEM* by double clicking onto the program icon or onto the icon of one of the picture or input text files.
The following chapters provide information about the menu items and dialog options. Some menu items can be selected by holding the Command key and pressing the key given at the menu name.

2.2 The *Apple* menu

About PlastFEM...

Like with any Mac OS program, the *About PlastFEM...* menu item gives you information on *PlastFEM*, e.g. the version number and the people involved in its development.

Other Apple Menu Items

You can use other programs or control panels selectable from the Apple menu as you would expect.

2.3 The *File* menu

New Graphics Window ⌘N
New Text Window
Open... ⌘O
Close ⌘W

These menu items apply to the text and graphics windows. *New Graphics Window* creates an empty graphics window to paste pictures into, *New Text Window* creates an empty text window where you can enter text to write a *PlastFEM* input file. and *Open...* lets you open an existing picture (type PICT) or text file (type TEXT, which need not necessarily be created with *PlastFEM*. Depending on the operating system version, the file can be selected either with the Standard File dialog (up to System 8.0) or with Navigation Services (Mac OS 8.5 and later).
The *Close* command lets you close the frontmost window, just like clicking in the window´s close box. If the window´s content has been modified since it was last saved, *PlastFEM* asks you if you want to save the changes, discard them, or cancel the *Close* command.
The drawing window is a special window which is automatically created when launching and cannot be closed. This is where all the FEM system drawing takes place.

Save ⌘S
Save As...

The *Save* and *Save As...* commands save the content of the frontmost window to disk. If you select *Save As...*, or if the content hasn´t been saved before, a dialog lets you choose a name and a folder to save the file to. *PlastFEM* generates standard PICT and TEXT files.

Page Setup...
Print... ⌘P

The *Page Setup...* command displays the standard Page Setup dialog of the selected printer that lets you specify printing options like the paper size. The *Print...* menu item lets you print the content of the frontmost window. You can select additional printing options like the number of copies from the standard Print dialog.

Quit ⌘Q

This command quits *PlastFEM* and returns to the Finder. If the content of some window was modified and hasn´t been saved, you will be asked whether to save the drawing, discard it, or cancel the *Quit* command. Before quitting, most *PlastFEM* options will be saved to the Preferences folder inside the System folder of the start-up disk.

2.4 The *Edit* menu

The *Edit* menu contains the standard Mac OS editing commands (*Undo*, *Cut*, *Copy*, *Paste*, *Clear*) and a menu item to specify *PlastFEM* preferences.

Undo ⌘Z

The *Undo* command only works for the drawing window. You can undo the last drawing operation with this command, but you can´t undo calculations. *Undo* simply deletes operations like *Draw System...* or *Draw Coordinates...* until the drawing window is empty.

Cut ⌘X
Copy ⌘C
Paste ⌘V
Clear
Select All ⌘A

For text and picture windows, these commands work as usual. For the drawing window, these commands deal with the complete drawing. You can *Cut* or *Copy* the drawing to the clipboard or *Clear* the drawing window. If you *Paste* a PICT from the clipboard, it gets added to the window content.

Preferences...

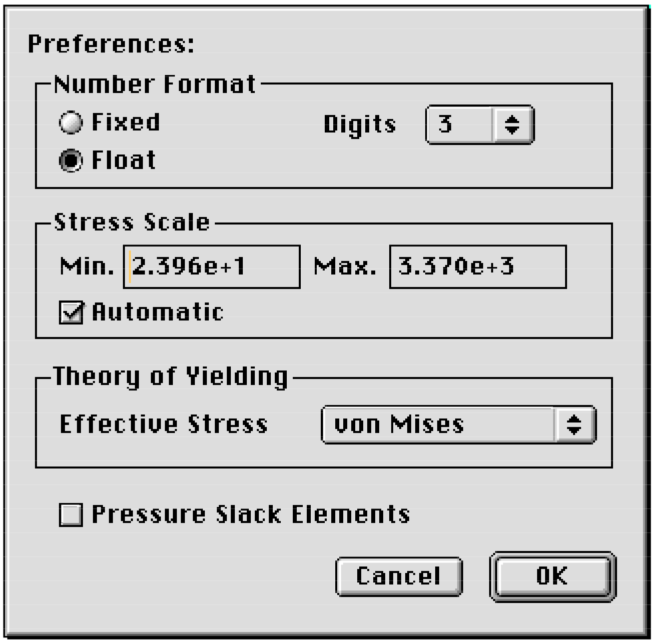


Fig. 2.1: Preferences Dialog

The *Preferences...* dialog gives you control about global program options.
You can change the format in which numbers are presented in dialogs and drawings. *Float* stands for scientific notation, e.g. 1.234e-1, and is most flexible to cover wide value ranges. *Fixed* lets you use decimal notation, e.g. 0.123. This is sometimes useful when drawing the coordinate system.
When stresses are drawn, you can display a color *Stress Scale*. The scale can be determined automatically by the program, or you can provide minimum and maximum values. This makes comparisons between load cases easier, but stresses outside of the predefined range will not be drawn.
PlastFEM currently only supports one theory of yielding, described in chapter 6, but you can select the *Effective Stress*, which specifies the stress state: *von Mises*, *Tresca* or *Normal Stress*. This selection will be used when the effective stress is drawn and when comparing stresses to the yield stress given in the problem description.
If *Pressure Slack Elements* is checked, the elements generated are not pressure resistant, like thin plastic foils that just wrinkle under pressure, for example. Elements, which are loaded by pressure, simply get a zero stiffness matrix. Since this presumes knowledge of the stress situation, an iterative calculation just like in the case of plastic material behaviour is necessary.

2.5 The *FEM* menu

Use this menu to control the finite element calculations. To complete a calculation, the menu commands basically have to be selected from top to bottom. It is possible to restart a calculation from every menu command in order to try different settings or to correct errors. You can quit *PlastFEM* at any stage without loosing results and resume calculations later. The graphic commands are available at any stage, too.
When you start a calculation for the first time, *PlastFEM* will prompt you to select a folder to store intermediate results. It will generate a few binary files that contain the stiffness matrix, the load forces and other data that can only be read by the program itself. Please consider creating a dedicated folder to avoid a jumble with other files. *PlastFEM* remembers this folder and the state of the problem after quitting, so you can continue with calculations at the same place when launching it again. To select the data folder a modified Standard File Dialog is used (see *Data Folder...*).
The problem to be analysed has to be described in specially formatted text files, which can be written using *PlastFEM* or any plain text editor. They may have any name you prefer and can be stored where you like. These files should not be put inside the data folder you selected to store intermediate results.
The problem definition files have to meet some specifications regarding their content.
Numerical values can be written in scientific or decimal notation and separated with spaces, commas, tabs or new lines. It doesn't matter how many values are in a line, just the number and order in a file is important.
You can include comments at any place in the file, starting with a slash. The rest of the line will be ignored. To ease navigating in a file, you can include numbers in parenthesis – "(" and ")" – at the beginning of a line. They will be ignored, too.
Consider this file as an example, where five numbers will be read (3, 1.2e5, 0.3, 100, -1000):

```
3   1.2e5, 0.3 / three numbers
(2) 100, -1000 / two more
```

If a file doesn't meet the required format, an alert like the one in Fig. 2.2 will be shown.

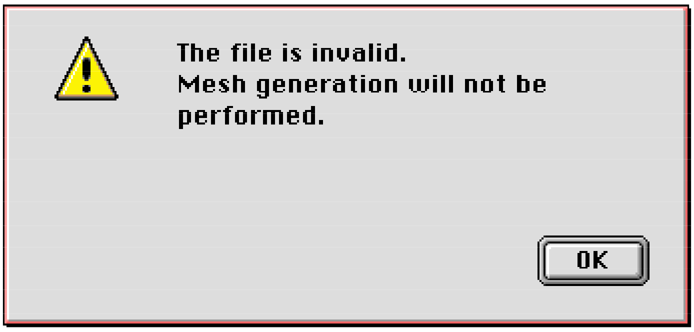


Fig. 2.2: Invalid file format dialog

Mesh Generator... ⌘0

PlastFEM is capable of generating a finite element mesh automatically, given only the outline of a sheet and the desired density. You have to define how many elements should be used for an outline (straight line or circular arc) and how they should be distributed. The mesh generator then builds elements, beginning from the outline and adding more layers until the whole area is meshed. It tries to generate numerically favourable elements (isosceles triangles). The method that is used here is based on Sadek[7].
It may be necessary to try a few different segmentations of the outline before the desired result is achieved, so *PlastFEM* draws the mesh as it is generated for immediate judging. Final corrections can be done by changing individual nodes (see chapter 2.10.3).
You can select a file with a Standard File dialog. This file has to be in the following format (comments are optional):

```
/ Mesh Generator Input Data
/
/ name of problem (one word)
Two.Areas
/
/ number of points
6
/ x and y point coordinates
0  0
100.0  0
100.0 100.0
0 100.0
200 0
200 100
/ number of closed areas
2
/ 1st area: Young's Modulus, Poisson's Ratio, Yield Stress,
/ Hardening Factor, Thickness
2.1e+5    0.30 10000 0  1
```

```
/ number of sides (area 1)
4
/ type, number of first point, number of elements, element ratio
0,  1 5 1
0, 2 5 1
0, 3 5 1
0, 4 5 1
/ end of first area
/
/ 2nd area: Young's Modulus, Poisson's Ratio, Yield Stress,
/ Hardening Factor, Thickness
2.1e+4    0.30 10000 0  1
/ number of sides (area 2)
4
/ type, number of first point, number of elements, element ratio
0, 2 5 1
0, 5 5 1
0, 6 5 1
0, 3 5 1
/ end of second area
/
/ number of links
1
/
/ link area/side to area/side
2 4 1 2
/
/ optimization yes/no (1/0)
0
/ end
```

The file starts with the title of the problem, which has to be a single word. Then the number of points and their x and y coordinates follow. These points are the corner or middle points of the area sides. The numbering of the points is arbitrary, it doesn't have to follow the side numbering. When the sides are defined, the points will be numbered in the order they appear in the input file.
The number of closed areas follows. An area is a closed region of the body that has to be meshed. It has some material properties and is described as a number of sides which consist of a starting point, the number of elements along the side and an element ratio, the size factor from one element to the next. The starting point of the following side is the end point of the side before. If the side is a circular arc, the center point has to be specified as well. The order of the sides is determined by mathematically positive circulation around the area (counterclockwise). The first side is arbitrary.
So for each area follows

- the material properties (Young's Modulus, Poisson's Ratio, yield stress, hardening factor and thickness) and
- the number of sides.

For each side follows

- the type of side (0 for a straight line, 1 for a convex, -1 for a concave circular arc),
- the starting point number,
- the center point number (only for arcs),
- the number of elements and
- the element ratio.

You can affect the meshing with the number of elements and the element ratio so that important regions of the body get a fine meshing, less important get a coarse meshing.

Then the number of links between the areas, resp. sides, follow and for each link the area and side numbers of the first and second area. This makes sure that adjacent elements at an area border share the same nodes and influence each other directly.

❖ The two sides belonging to a link have to have the same number of elements.

An option for the automatic optimization of the node numbers concludes the input file. By building the mesh in layers, a big stiffness matrix is necessary, due to the resulting big numbering differences of nodes inside an element, if no renumbering is performed.

The matrix structure gets calculated after the meshing is complete, and the size of the matrix is displayed in a dialog (Fig. 2.3).

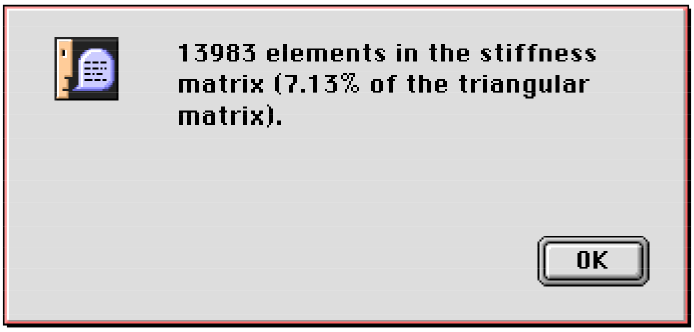


Fig. 2.3: Information about the global stiffness matrix

The example file above results in the mesh shown in Fig. 2.4.

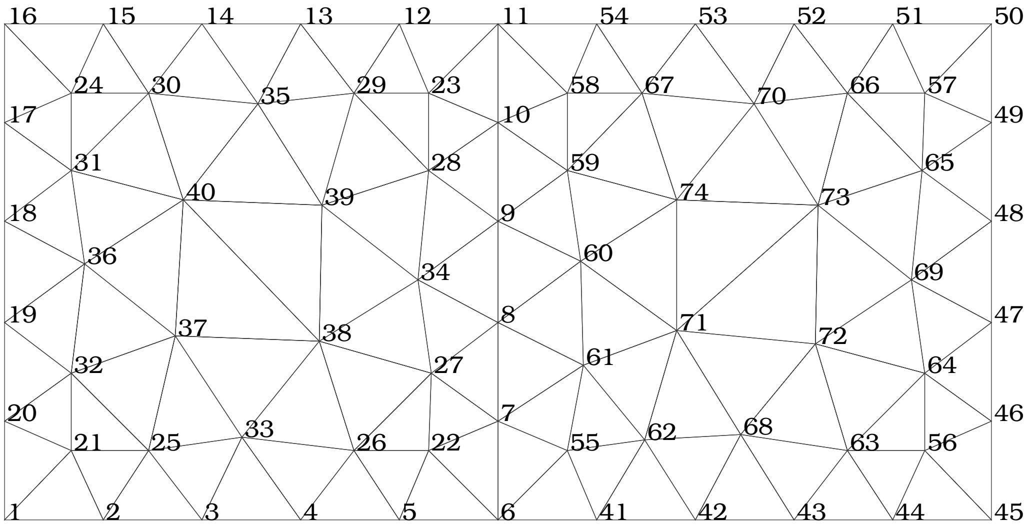


Fig. 2.4: Example of a meshing (two areas)

You can even generate bodies with holes this way. Two or more areas around the hole have to be defined and linked. Two sides of the same area can be linked as well, so you can use just one area.

The following listing of a mesh generator file and Fig. 2.5 demonstrate a punched body.

```
/ Mesh Generator Input Data
/
/ name of problem (one word)
Punched
/
/ number of points
8
/ x and y point coordinates
(1) 0 0
(2) 20 0
(3) 6 6
(4) 14 6
(5) 6 14
(6) 14 14
(7) 0 20
(8) 20 20
/ number of closed areas
2
/ 1st area: Young's Modulus, Poisson's Ratio, Yield Stress,
/ Hardening Factor, Thickness
2.1e+5    0.30 10000 0 1
/ number of sides (area 1)
6
/ type, number of first point, number of elements, element ratio
0 1 10 1
0 2 4 1
0 4 4 1
0 3 4 1
0 5 4 1
0 7 10 1
```

```
/ end of first area
/
/ 2nd area: Young's Modulus, Poisson's Ratio, Yield Stress,
/ Hardening Factor, Thickness
2.1e+5    0.30 10000 0  1
/ number of sides (area 2)
6
/ type, number of first point, number of elements, element ratio
0 2 10 1
0 8 10 1
0 7 4 1
0 5 4 1
0 6 4 1
0 4 4 1
/ end  of second area
/
/ number of links
2
/ link area/side to area/side
1 2 2 6
1 5 2 3
/ optimization yes/no (1/0)
1
/ end
```

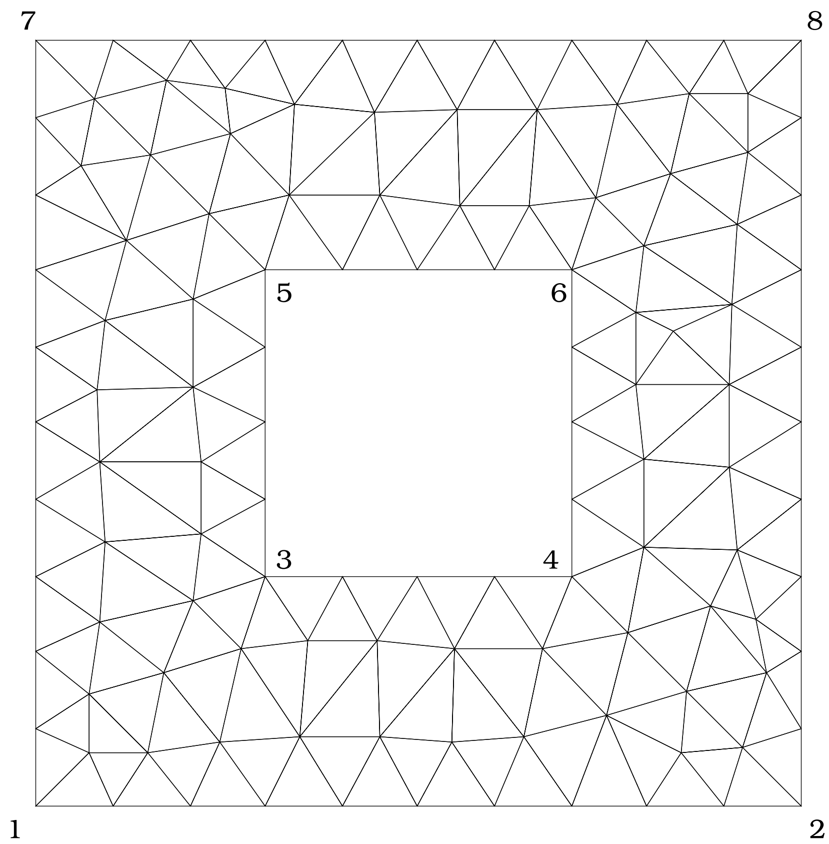


Fig. 2.5: Punched body

The areas are linked at the lines between point 2 and 4 and between point 5 and 7.

The node numbering is independent from the point numbering. The nodes are numbered as the areas get filled by elements, from the border to the middle and one area after the other (Fig. 2.4). The automatic optimization of the node numbers distributes the node numbers across all areas.
Here is an example for a circle area (Fig. 2.6):

```
/ Mesh Generator Input Data
/
/ name of problem (one word)
Disc.Mesh

/ number of points
3
/ x and y point coordinates
0  0
0  5
0 10
/ number of closed areas
1
/ 1st area: Young's Modulus, Poisson's Ratio, Yield Stress,
/ Hardening Factor, Thickness
2.1e+5    0.30 10000 0 1
/ number of sides (area 1)
2
/ type, number of first point, number of elements, element ratio
1, 1 2 14 1
1, 3 2 14 1
/ end of first area
/
/ number of links
0
/ optimization yes/no (1/0)
0
/ end
```

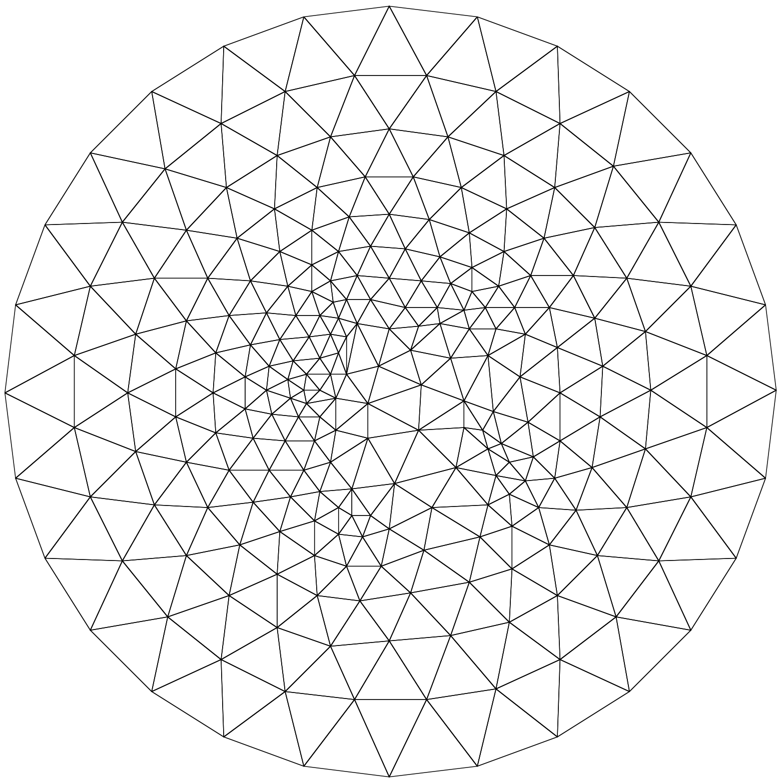


Fig. 2.6: Disc mesh

Geometry...   ⌘1

The geometrical data for an already existing mesh can be read in from a file with this menu command. In contrast to a mesh generator data file, all nodes and elements have to be defined in a geometry data file. It has the following format:

```
/ Geometry Data with 8 elements and 9 nodes
/ name of problem (one word)
Small.Sample
/
/
/ number of nodes
9
/ nodal coordinates
0   -100
0         0
0    100
100 -100
100    0
100  100
200 -100
200    0
200  100
/
/ total number of elements
8
```

```
/ number of elements (first area)
8
/ Young's Modulus, Poisson's Ratio
2.1e5, 0.3
/ Yield Stress,
500
/ Hardening Factor, Thickness
0.01  1
/
/ nodes of elements
1 4 5
1 5 2
2 5 3
3 5 6
5 8 6
4 8 5
4 7 8
6 8 9
/ end
```

The file starts with the name of the problem (one word), then the number of nodes, the coordinates of all nodes and the total number of elements follow. For each area (which can differ from others by material properties) you have to specify the number of elements, Young's Modulus, Poisson's Ratio, yield stress, hardening factor and thickness, as well as the node numbers of each element.

❖ The assignment order of the global node numbers to the local numbers (1,2,3 - Chap. 4) has to be counter-clockwise.

The example above looks like Fig. 2.7:

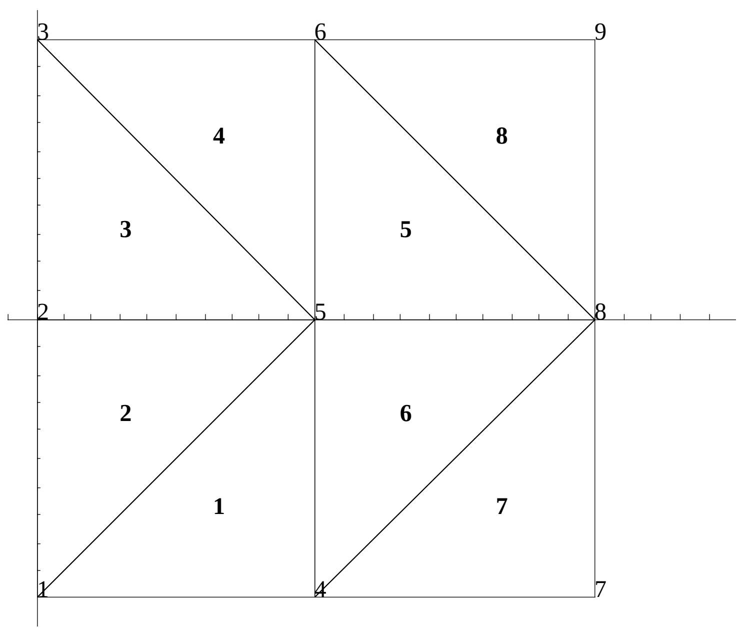


Fig. 2.7: Small sample

After reading the data, the matrix structure gets calculated and the mesh can be displayed.
The main use of this command is to read in structures that have been generated by the mesh generator but altered after the generation, for example by graphically changing nodal coordinates or by adding or deleting elements.

Stiffness Matrix ⌘2

This menu command is enabled when the geometrical and material data is known from reading a mesh generator or geometry data file. The element stiffness matrices are calculated and sorted into the global stiffness matrix. During this operation, a window shows the current element number.

Support... ⌘3

Using a Standard File or Navigation Services dialog, you have to select a support data file with which has the following format (comments are optional, again):

```
/ Support Input Data for Small Sample
/
/ number of restricted displacements
3
/
/ direction (Tx, Ty or Ta), node, value of displacement
Ta 7 0.0
Ta 8 0.0
Ta 9 0.0
/
/ end
```

Each node has two degrees of freedom, that means two possible directions to move (Translation in x and y direction). Use Ta as keyword to restrict all Translations of a node.
At least three degrees of freedom have to be restricted in a finite element system in order to prevent rigid body motion and to allow for the solution of the equation system. Both coordinate directions must be included, for example by fixing one node and setting the *x* displacement of another node to zero.
In the example above three nodes have been fixed (Fig. 2.8).

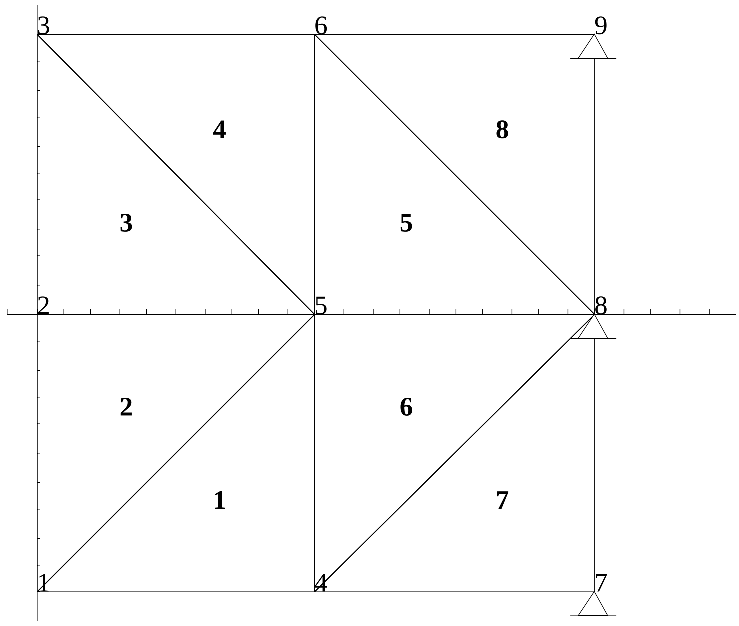


Fig. 2.8: Support sample

After reading the support data, the stiffness matrix gets rearranged into L^TDL form to prepare for the solution (refer to chapter 3). During the calculations, the current row of the stiffness matrix is shown in a window.

Load... ⌘4

The load data file is similar to the support data file. It has the following form:

```
/ Load Input Data for Small Sample
/
/ number of loads
4
/
/ direction, node and value of load
Fx 1   -250
Fx 2   -250
Fy 3   500
Fy 6   500
/
/ ---- input data for nonlinear calculations ----
/
/ number of load steps, max. number of iterations, residual in %
50 50 0.1
/
/ number of displacements to log (0 = no log)
2
/
/ direction, number of node
Tx 3, Ty 3
/ end
```

The forces are defined by direction (Fx or Fy), number of node and value. Distributed loads have to be converted to concentrated loads at the node points (please refer to chap. 3.2).

The load data file holds additional information that is necessary for nonlinear calculations. That is, the number of load steps that have to be executed, the maximum number of iterations per step and the acceptable size of the residual in percent. These values must always been given, even on linear problems, to make the load file complete.

It is possible to log displacements during nonlinear calculations. If this option is chosen, a text file *Log.text* is written to the data directory. It contains a table of the load stage in percent and the displacement(s). This table can be imported into a spreadsheet program for graphical analysis, for example. If the number of displacements to log is zero, no log file is generated.

After reading the load data, *PlastFEM* solves the linear equation system to calculate the nodal displacements. Subsequently, the strains and stresses are calculated and the user is told whether there will be plastic deformations or not (Fig. 2.9).

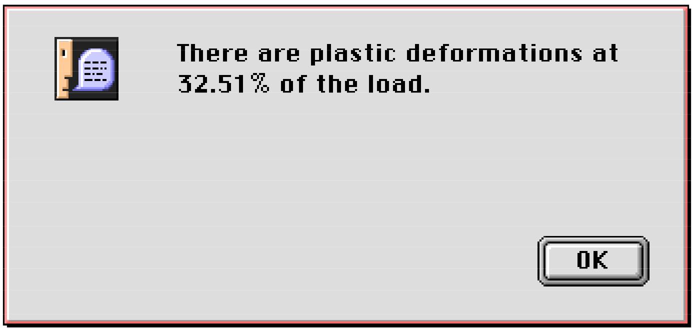


Fig. 2.9: Information about plastic deformations

If the yield point is crossed and thus plastic deformation takes place, the load is scaled down so that the maximum effective stress equals the yield stress. From that point on, nonlinear calculations have to be performed.

When the *Load* step is done, the finite element system can be drawn with displacements and stresses.

Nonlinear Calculation... ⌘5

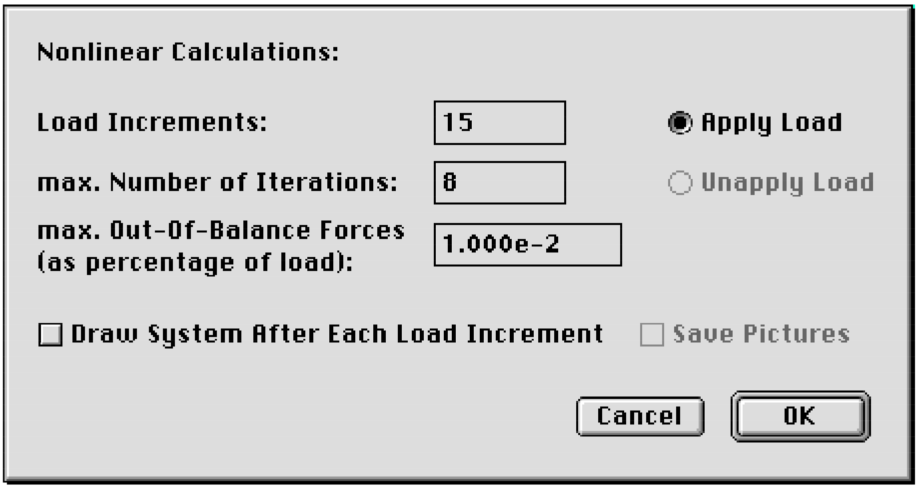


Fig. 2.10: Nonlinear calculation dialog

In this dialog you can specify parameters for the nonlinear calculations that are necessary for elasto-plastic material behaviour or pressure slack elements. The input fields for the load increments, the iterations and out-of-balance forces (residual) are already filled with the values of the load data file, but these values can be altered here.
At each load step, the program iterates the problem state until the acceptable size of the residual or the maximum number of iterations is reached. The residual is build by adding the squares of all out-of-balance forces (chap. 5), and it gets compared to the sum of the squares of the external loads.
The difference between the load at yield stress and the total load gets divided into equal load steps. The number of load increments should be chosen so that the quotient of load step size and elastic load becomes 0.1 to 0.3. The residual should be between 0.1 and 1 percent.
When the load is applied, you can stop the calculations by pressing command-period (⌘-.).

❖ In order to repeat the load calculation, for example to test different settings, you have to start with the *Load* command and select a load file again, so that the elastic calculations are performed first.

During the nonlinear calculations, intermediate results can be displayed in the graphics window and these pictures can be saved. The FE system is drawn with the current options of the *Draw System* dialog. This allows you to watch the development of the yield zone, the stresses and displacements. There has to be an

open graphics window, of course. To save the pictures after each load step, the graphics window has to be saved once before you start the calculation, in order to define a path name (a number gets appended to the name for each step).
At the end of the calculations, the highest number of iterations that was needed is displayed. If the demanded accuracy could not be achieved, the residual gets displayed and can be used to adjust the load step or the maximum number of iterations.
When an elasto-plastic body gets partly plastic during deformation, some deformations and stresses remain even after the load is removed. This behaviour can be simulated by *PlastFEM* if you select the *Unapply Load* option when choosing *Nonlinear Calculations* a second time. The entire load is removed in one elastic step.

❖ You should not change the number of load increments between applying and unapplying the load. Otherwise the forces can not be scaled properly.

System Information... ⌘I

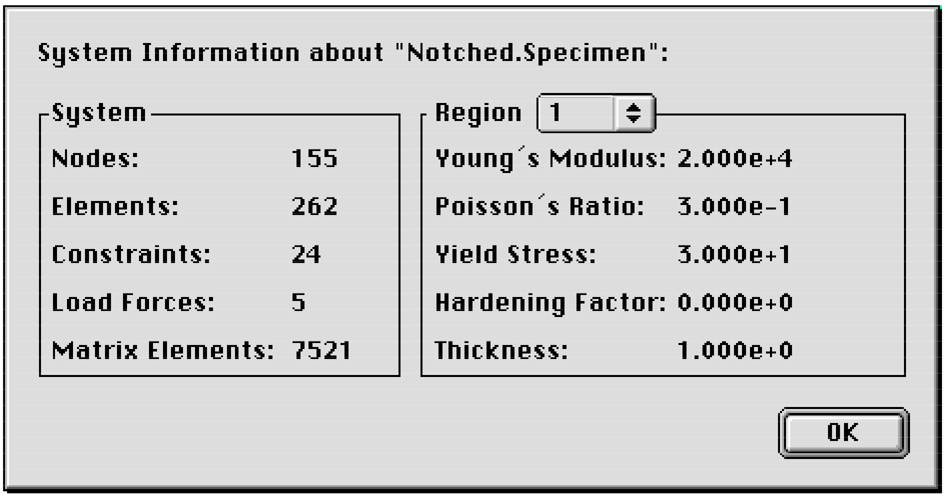


Fig. 2.11: System information dialog

By selecting this menu command, a dialog with information about the current problem is displayed (Fig. 2.11). If there is more than one area, you can get the material properties of any area by selecting the number from the pop up menu.

Output...

Geometrical data and calculation results can be written to a text file or printed. The text files are readable by any text editor. The available options are shown in Fig. 2.12.

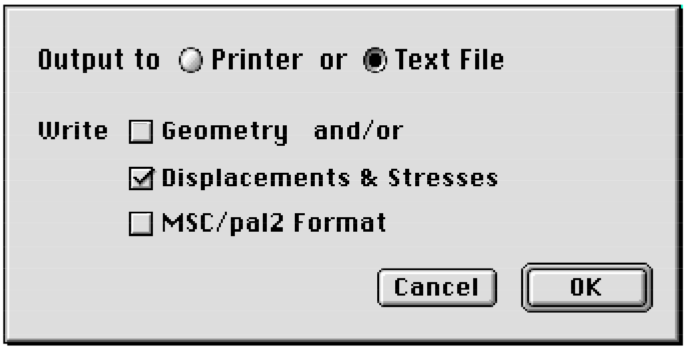


Fig. 2.12: Output dialog

When a mesh has been manually altered, it is easier to save the altered structure as a geometry input data file than to repeat the manipulations after performing the *Mesh Generator* step. When *MSC/pal2 Format* is chosen, *PlastFEM* changes the format of the output file to match the input format of a *MSC/pal2* geometry file.

Data Folder..

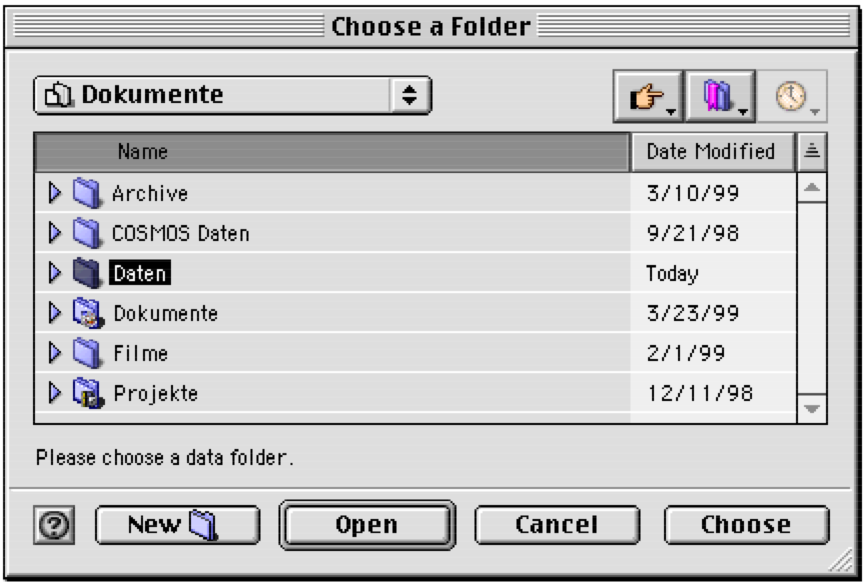


Fig. 2.13: Data folder dialog

Intermediate calculation results are saved in a few files inside a data folder. This folder has to be chosen before any calculations can take place. You should choose a separate folder for this purpose, not the program or the input file folder, as this folder gets crowded by data files. But you can quit *PlastFEM* at any stage of the calculations and resume later because of the saved results.

2.6 The *Graphics* menu

Draw System... ⌘D

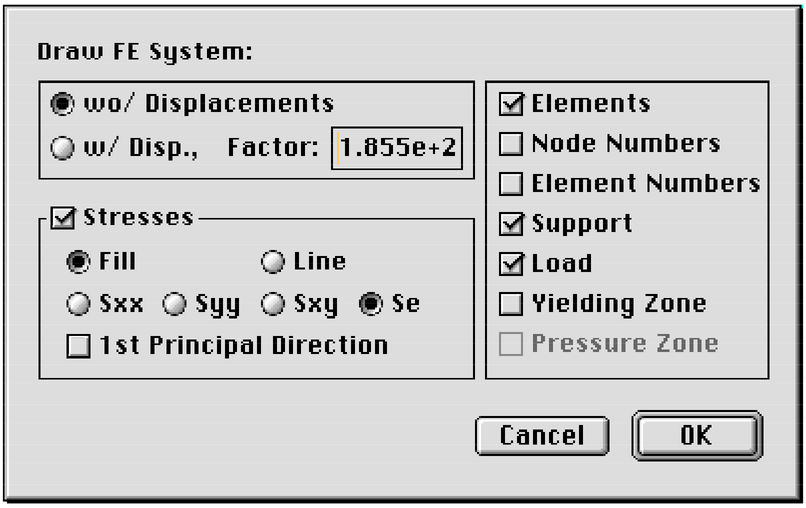


Fig. 2.14: Dialog for drawing the finite element system

PlastFEM supplies a few options to display the finite element system, but their availability depends on the stage of the calculations and the nature of the problem.

Before the calculations, the element mesh can be drawn undeformed and with or without node and element numbers. This is useful for checking the geometrical data. For all other options, calculations have to be done.

When drawing the mesh with displacements, a scale factor has to be chosen to make the displacements visible and to display them in a reasonable proportion to the body. *PlastFEM* suggests a suitable factor according to the last calculation.

The stresses are drawn as color lines of constant stress or as filled regions. Each color represents a certain stress (when drawing lines) or stress section (when drawing regions). Low stress values are green, high values are red, and there are 12 colors in between. An absolute high negative stress value gets drawn in green when there are more positive values. The assignment between stresses and colors can be seen in the *Stress Scale* window which can be opened by selecting the *Stress Scale* menu command from the *Graphics* menu.

You can display the normal stresses σ_{xx} and σ_{yy}, the shear stress σ_{xy} or the effective stress σ_e. The direction of the first principal stress can be drawn as well; it is shown as a small line inside each element. When stresses are drawn, you can decide whether to draw the element borders or not. The stress distribution of a body with many elements can be seen more clearly without the element borders.

The constraints are represented by small moving or rigid support symbols, depending on how many degrees of freedom of a node are restricted. External loads are drawn as arrows.

The yielding zone, in which the plastic deformation takes place, is drawn in black, and the pressure zone (when using pressure slack elements) is drawn in grey.

Drawing Area... ⌘B

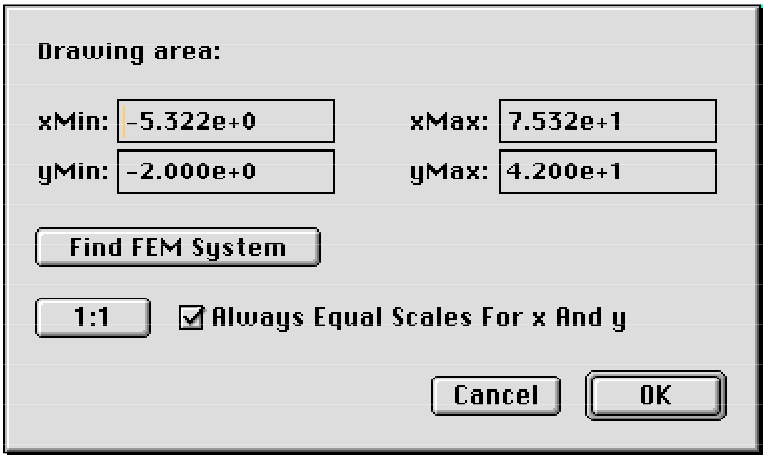


Fig. 2.15: Drawing area dialog

The *Drawing Area* command lets you choose the borders or the displayed area. You can specify the values directly by entering them into the edit fields, or you can click on *Find FEM System* to let *PlastFEM* search the extremes of the coordinates of the current problem.

The *1:1* button adjusts the corresponding maximum coordinate to the current size of the drawing window so that both directions have the same display ratio. If *Always Equal Scales for x and y* is checked, this ratio is automatically adjusted when the size of the drawing window is changed. You have to redraw the FEM system then.

Coordinate System... ⌘K

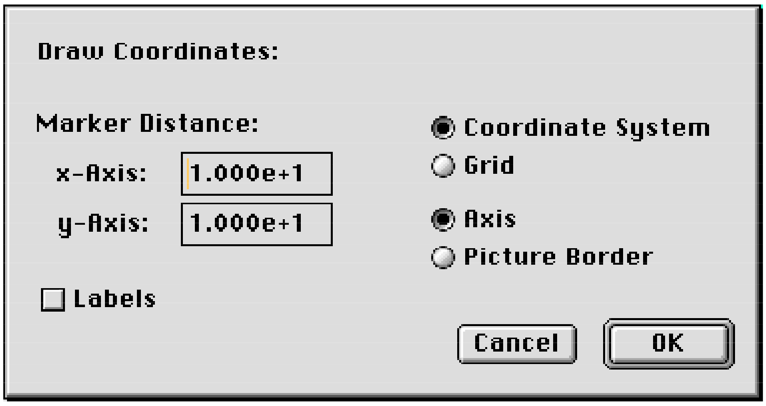


Fig. 2.16: Coordinate system dialog

The coordinate axes can be drawn with or without labels. You specify the marker distance by entering the chosen values into the edit fields. If *Axis* is selected, the axes go through the coordinate origin, otherwise the system is drawn at the *Picture Border*.
When you draw a *Grid* instead of a *Coordinate System*, the marker distance equals the grid distance.

Stress Scale

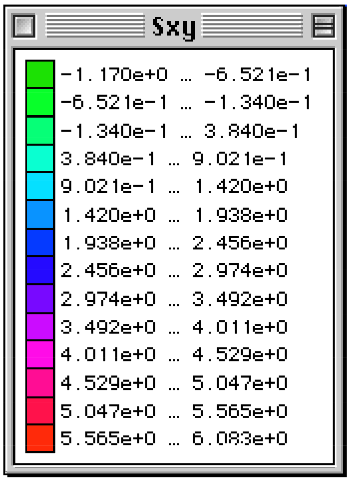


Fig. 2.17: Stress scale

This command shows a window with the assignment between stresses and colors. You can compare these colors to the colors in the picture to see how high the stress is in a certain area. When a different kind of stress is drawn, the stress scale is adjusted accordingly.

To include the stress scale as part of the picture in the graphics window, you can move the scale window to a place where you like it to be drawn and click on it while pressing the command key. The stress scale then gets copied to the selected place.

2.7 The *Pen* menu

Black
Red
Green
Blue
Cyan
Magenta
Yellow

These commands select the color of the pen that is used when drawing the elements, the coordinate system and the grid.

0.25 Point
0.5 Point
1 Point
2 Point
3 Point

These commands control the thickness of the drawing pen. You can´t tell the difference between the 0.25, 0.5 and 1 Point settings on the screen, but it is noticeable when the picture is printed.

Grey

Use a grey mask when drawing. This is useful to indicate the original mesh behind the deformed mesh, for example, or to draw the grid.

2.8 The *Text* menu

Font
Style
Size

The *Text* menu has the common submenus *Font*, *Style* and *Size*. *PlastFEM* supports all fonts that are installed in the System folder and provides commands for the usual styles and sizes. When the drawing window is frontmost, the font you choose will be used to label the coordinate system and the stress scale, when a text window is frontmost, the font will be used for the entire text.

2.9 Graphics options

2.9.1 Element information

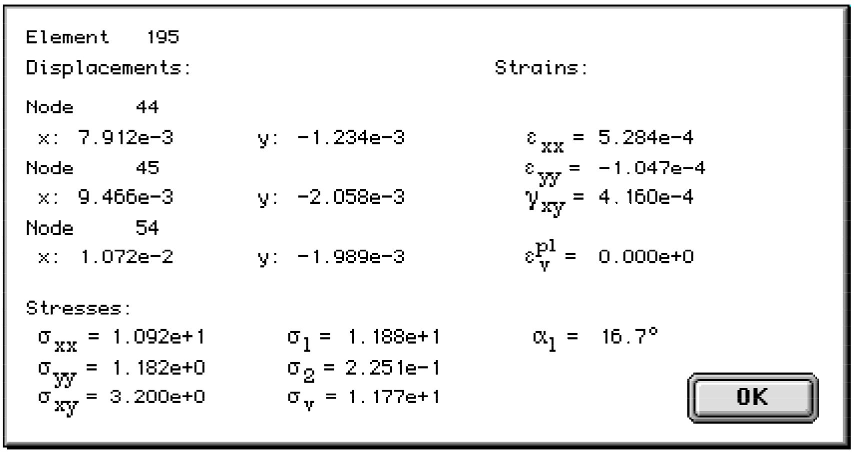


Fig. 2.18: Information about an element

If the linear equation system has been solved, you can get information about an element by double clicking on it. An alert window like in Fig. 2.18 shows the displacements of the nodes of the element, the strains and the stresses in the element:
σ_{xx} and σ_{yy} are the normal stresses in a plane perpendicular to the respective coordinate axes, ε_{xx} and ε_{yy} the strains in the respective directions. σ_{xy} is the shear stress in these planes, which is equal to σ_{yx}. γ_{xy} is the alteration of the right angle between two line elements parallel to the coordinate axes.
σ_v is the effective stress according to the settings in the *Preferences* dialog (see Chap. 2.4), ε_v^{pl} is the effective plastic strain in the element.

The stresses σ_1 and σ_2 are the principal stresses, and α_1 is the angle between the x-axis and the first principal stress direction.

2.9.2 Selecting a drawing area

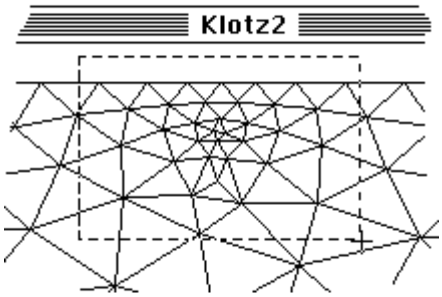


Fig. 2.19: Selecting a drawing area

You can select a drawing area by clicking into the drawing window and dragging a marquee. After releasing the mouse button, a dialog gives you the option to make the selected area active or to cancel.

2.9.3 Moving of nodes

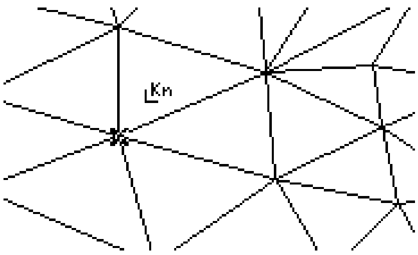


Fig. 2.20: Moving a node

If you click on a node when holding down the option key, the node is selected and indicated by a small cross. The cursor changes to suggest that you can move the node by clicking onto another place.
The finite element system first has to be drawn by selecting the *Draw System* command before you can move the nodes, because when the mesh is drawn during the generation, not all necessary information is saved.
The mesh isn't redrawn automatically when a node is moved, you have to do this by pressing *option-command-D* or by selecting *Draw System* from the *Graphics* menu.

When you finished changing node positions, you should save the changed geometry to a file by selecting *Output* from the *FEM* menu.

2.9.4 Finding node numbers

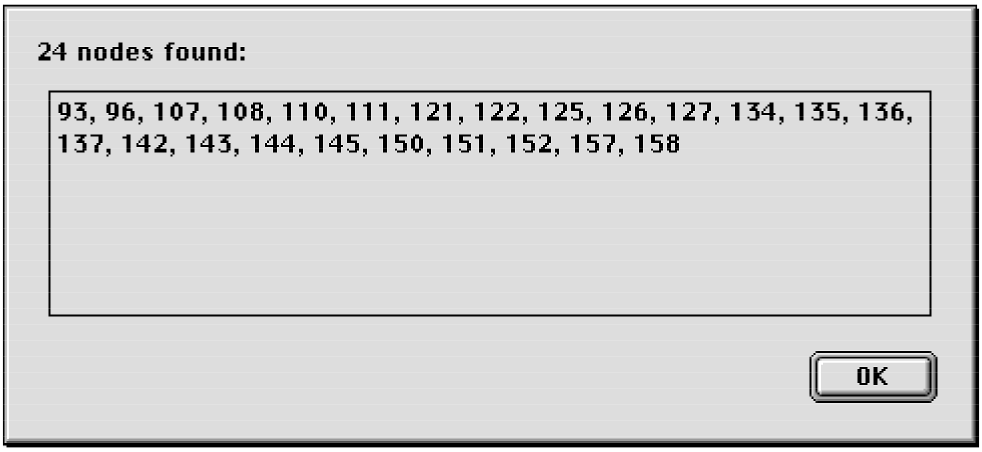


Fig. 2.21: Display of node numbers

You can see the numbers of nodes by selecting the *Node Numbers* option in the *Draw System* dialog and draw the FE mesh, but you can get the numbers of certain nodes by pressing the command key and dragging the marquee around them as well. A dialog (Fig. 2.21) will show you the numbers and the nodes will be indicated by small crosses.
This is especially useful for building the Support and Load files, as the optimization that the mesh generator performs leads to non-consecutive node numbering on the sides of a body.

2.9.5 Drawing Window size

The drawing window size can be changed by dragging the size box of the window. If there is a drawing inside the window already, it is scaled to the new size. The scaling causes loss of details and distortion, so you should clear the window and draw it again if you want better quality.

3. Example problems

The input data files of the following examples are located in the folders with the respective names inside the *Input Files* folder of the *PlastFEM* distribution. If you use *PlastFEM* for editing these files, be sure to save the changes before using them for calculations.

Each example has four files which are named according to their functions: The names of the input files for the mesh generator end with *Mesh*, and the geometry files are named *Geometry*, just like the *Support* and *Load* files. The files don´t have to have special names, but it is easier to tell them apart that way.

You can try the example problems with the *Geometry*, *Support* and *Load* files by selecting the commands 1 to 5 from the FEM menu in a row. The *Geometry* files contain the slightly changed mesh that was generated by the mesh generator. If you want to see the mesh generation, you can use a *Mesh* file instead of the *Geometry* file and start with the *Mesh Generator* command, leaving out the *Geometry* command. If there aren't any plastic deformations in an example, you can't select the *Nonlinear Calculation* command.

Triangular elements with three nodes and linear interpolation functions give constant stresses in the element area (see Chap. 4). This would result in a single color in every element when stresses are displayed. In order to get a more realistic display of the stresses, *PlastFEM* calculates the average of the stresses of all elements connected to a node and generates a stress plane over each element. This plane is marked with all the colors that correspond to the stress area between the node averages. Lines of equal stresses can be drawn or the area between two lines is filled, depending on the drawing options.

3.1 Force Distribution

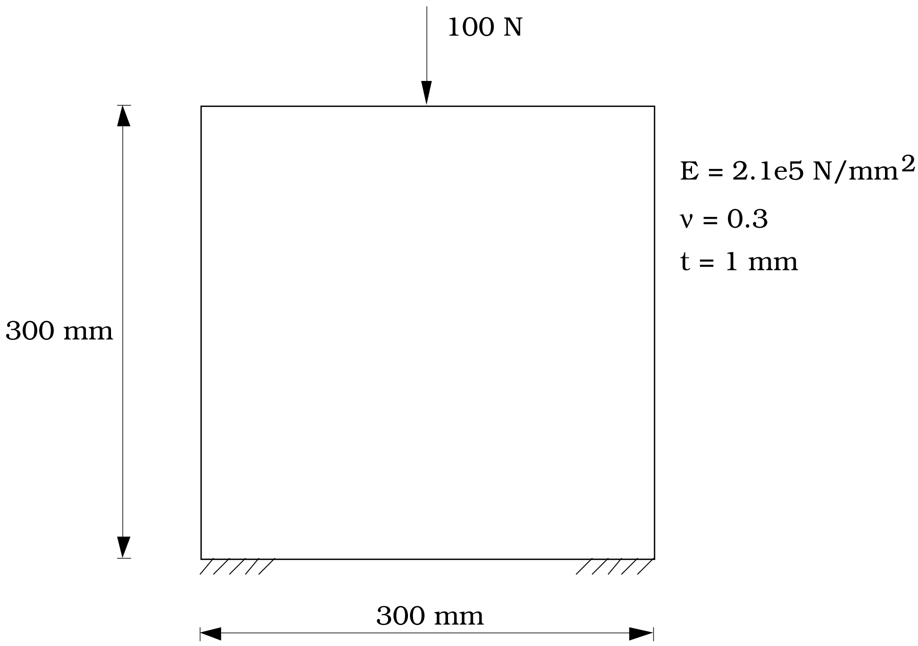


Fig. 3.1: Force distribution example

This example illustrates the rapid fading of stresses from the load point of a single force. Fig. 3.2 shows the lines of equal effective stress. The lines have the same difference of magnitude to each other, so you can tell the high gradient in areas with dense lines.

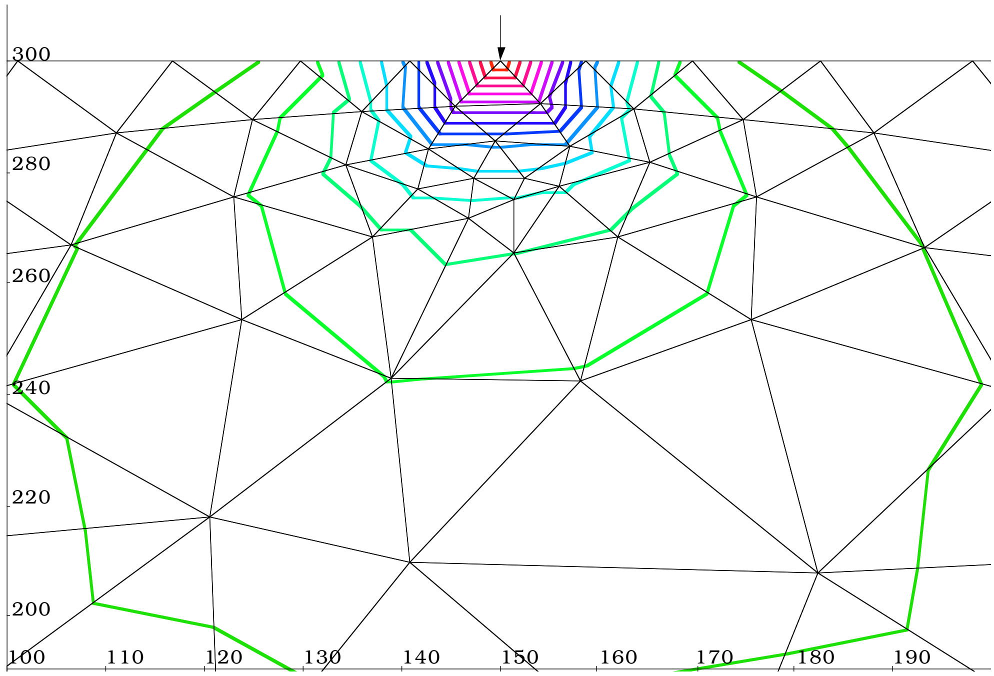


Fig. 3.2: Lines of equal effective stress (force distribution example)

Areas with a high stress gradient should be divided by more and smaller elements for higher accuracy and in order not to miss a stress peak. Therefore, the upper edge of the example has been divided into two lines to prepare for the mesh generation. The first line has an element ratio of 0.83, so that the elements get smaller in the direction towards the load point. The second line's ratio is the reciprocal value 1.2, so that the distance between the second and third node on the line is 1.2 times the distance between the first and second node: The elements get bigger.

```
/ Mesh Generator Input Data
/
/ name of problem (one word)
Force.Distribution
/
/ number of points
5
/ x and y point coordinates
0.0  0.0
300.0  0.0
300.0 300.0
150.0 300.0
0.0 300.0
/ number of closed areas
1
/ Young's Modulus, Poisson's Ratio
 2.1e+5    0.30
/ Yield Stress, Hardening Factor, Thickness
100 0.01  1
/ number of sides
5
/ type, number of first point, number of elements, element ratio
0 1 10 1   / bottom
0 2 10 1   / right
0 3  8 0.83   / top right
0 4  8 1.2   / top left
0 5 10 1   / left
/
/ number of links
0
/ optimization yes/no (1/0)
1
/ end
```

3.2 Square Angle

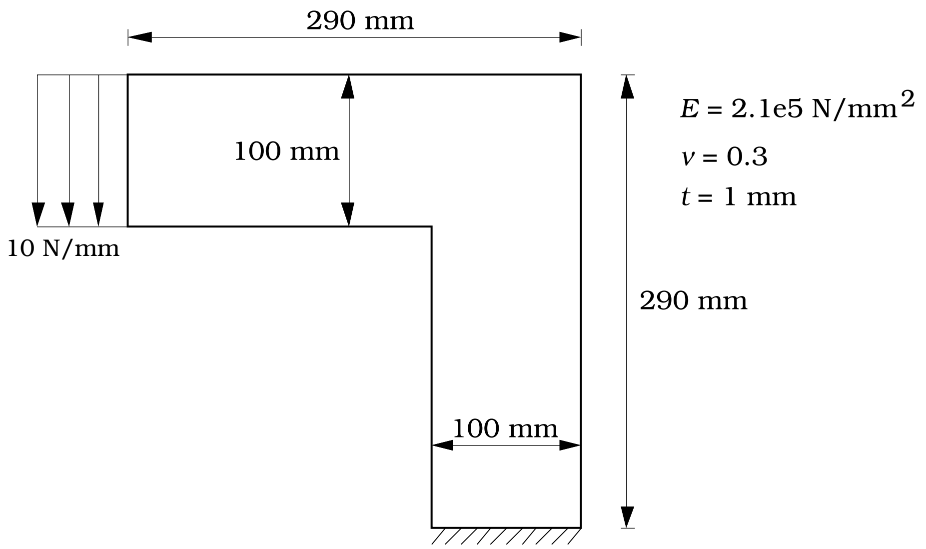


Fig. 3.3: Square angle

The sheet-metal square in Fig. 3.3 is bent by a distributed load of 10 N/mm. This load has to be transformed into statically equivalent node forces, since only nodes can be loaded. The left end of the upper side is divided into three elements, so each element carries a load of

$$\frac{10\,\mathrm{N/mm} \cdot 100\,\mathrm{mm}}{3} = 333{,}33\,\mathrm{N}.$$

This value has to be equally divided onto the two border nodes (166,67 N for each node). Finally, we have to sum up the load for each node: The nodes on the outside get a force of 166,67 N, the one in the middle gets 333,33 N.
Fig 3.4 shows the deformed angle (displacements enlarged by factor 20) with the effective stress. You can see the stress concentration at the notch.

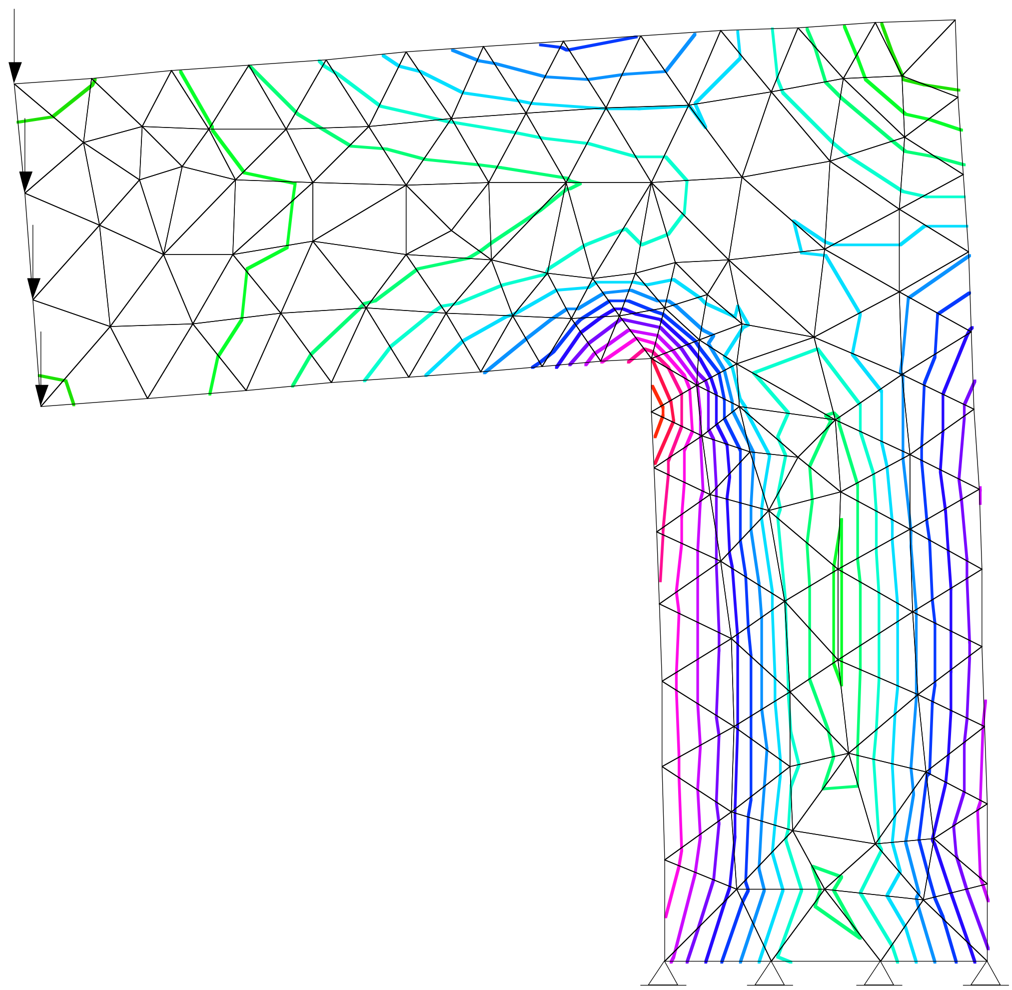


Fig. 3.4: Deformed square angle with effective stress

3.3 Perforated Plate

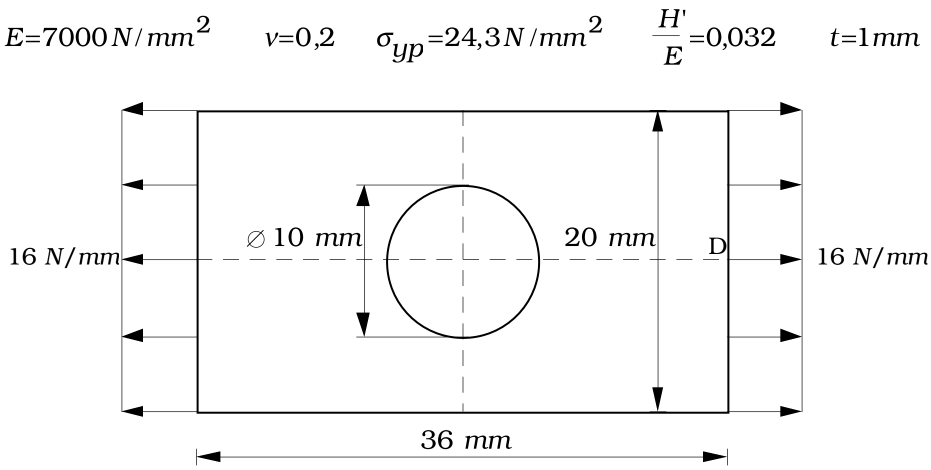


Fig. 3.5: Perforated plate

This example shows a plate with a hole under uniaxial load. The load is high enough to exceed the yield stress, so that plastic deformation will take place. Since the body is symmetrically in respect to two directions, only one quarter has to be examined. The displacements of the nodes along the symmetry lines have to be restricted perpendicular to the lines to get the same conditions as in the complete model. In this example, the lower right quarter has been chosen (Fig. 3.6).

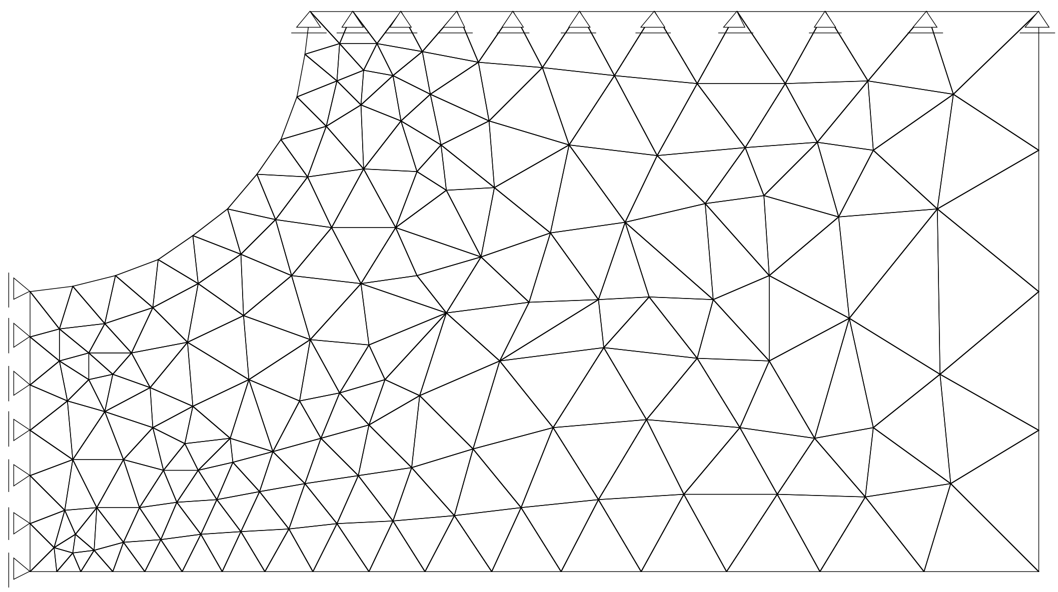


Fig. 3.6: Discretized lower right quarter of symmetric plate

During the load process from the first appearance of yield stress to the maximum load, three areas with plastic deformation will develop, and two of them will grow together. To watch the development you can select *Draw System After Each Load*

Increment from the *Nonlinear Calculation* dialog. Before executing the nonlinear calculations, you should draw the system once with the desired options, preferably after the elastic calculations and with *Yielding Zone* and *Stresses* selected.
The following pictures (Fig. 3.7 to Fig. 3.9) show the state of the problem at selected load increments. The load process is divided into 15 increments and the displacements are enlarged by factor 20.

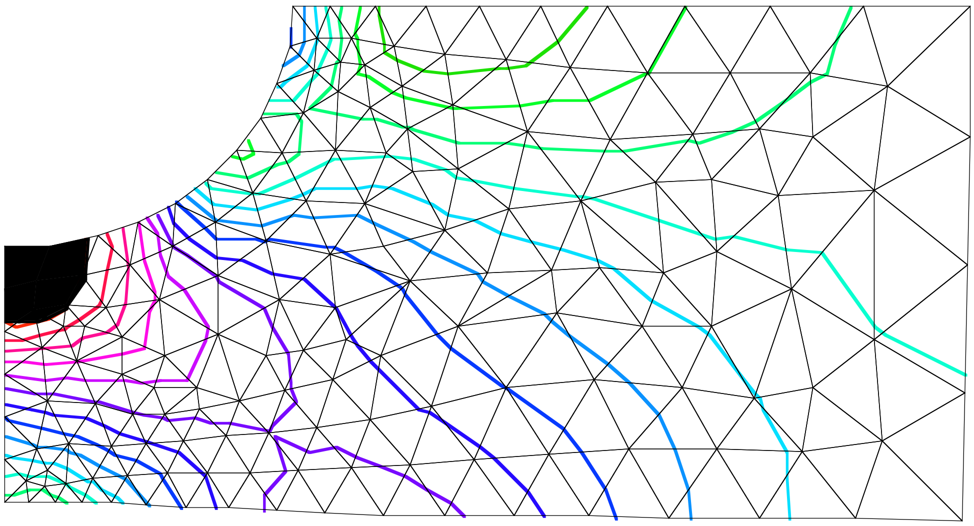


Fig. 3.7: 5th load increment

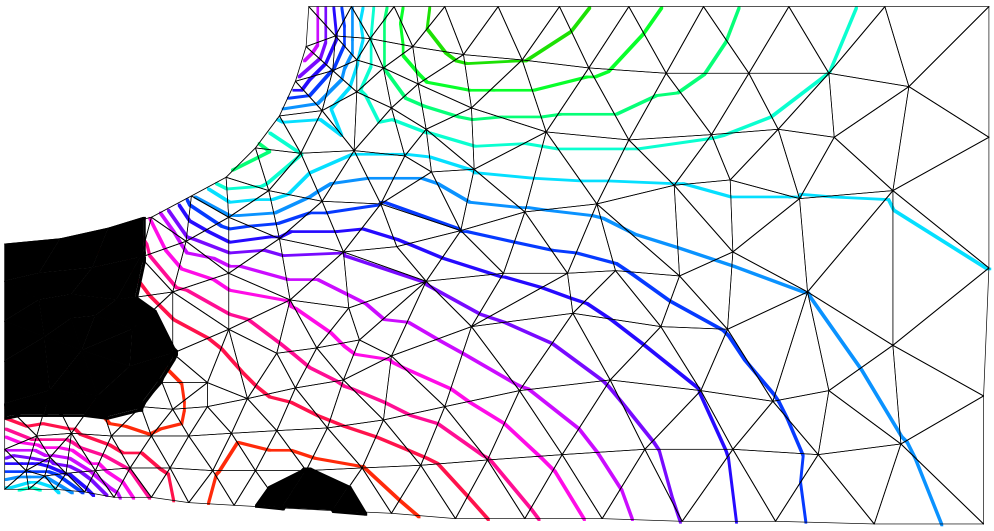


Fig. 3.8: 10th load increment

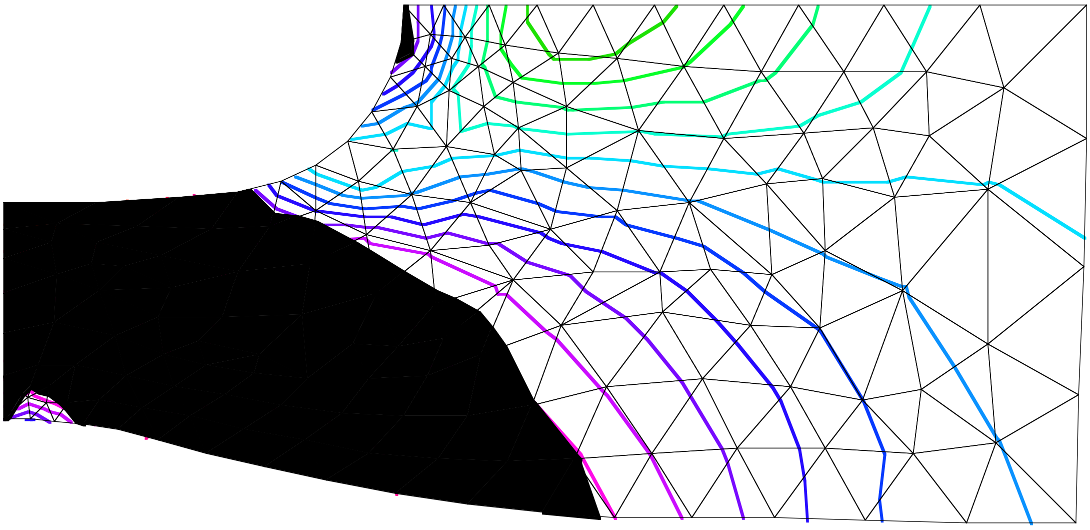


Fig. 3.9: 15th load increment

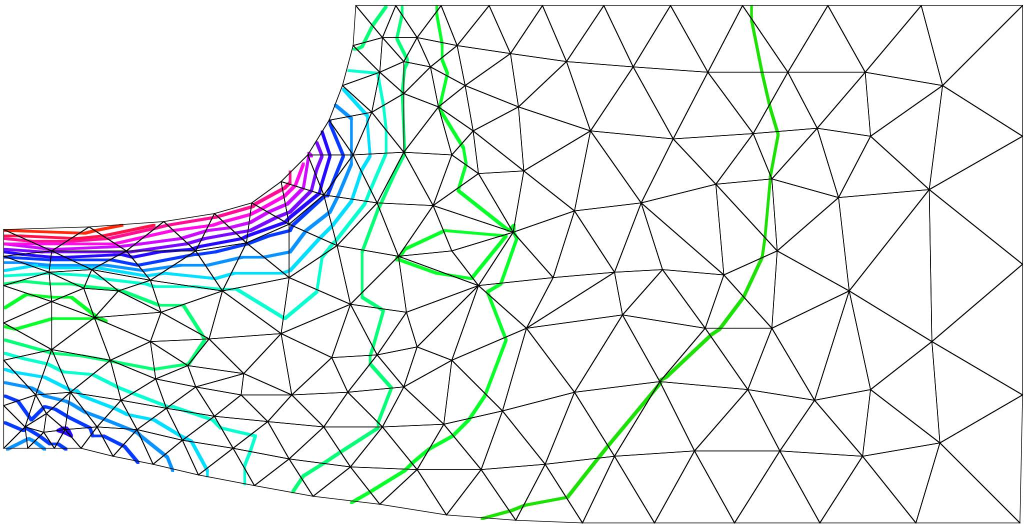


Fig. 3.10: Effective stress distribution after detaching the load

At full load, almost the complete cross-section at the hole has been deformed plastically. Since the deformation is not uniform across the body, there are still stresses after the load has been detached (Fig. 3.10). You can simulate the detachment of a load by doing nonlinear calculations again, this time selecting *Unapply Load*.

You can log certain displacements during nonlinear calculations with *PlastFEM*. In this example, the horizontal displacement of point *D* in Fig. 3.5 (node number 168) has been logged. The results have been written into a file (*Log.text*) by *PlastFEM* and imported into a spreadsheet program (Fig. 3.11).

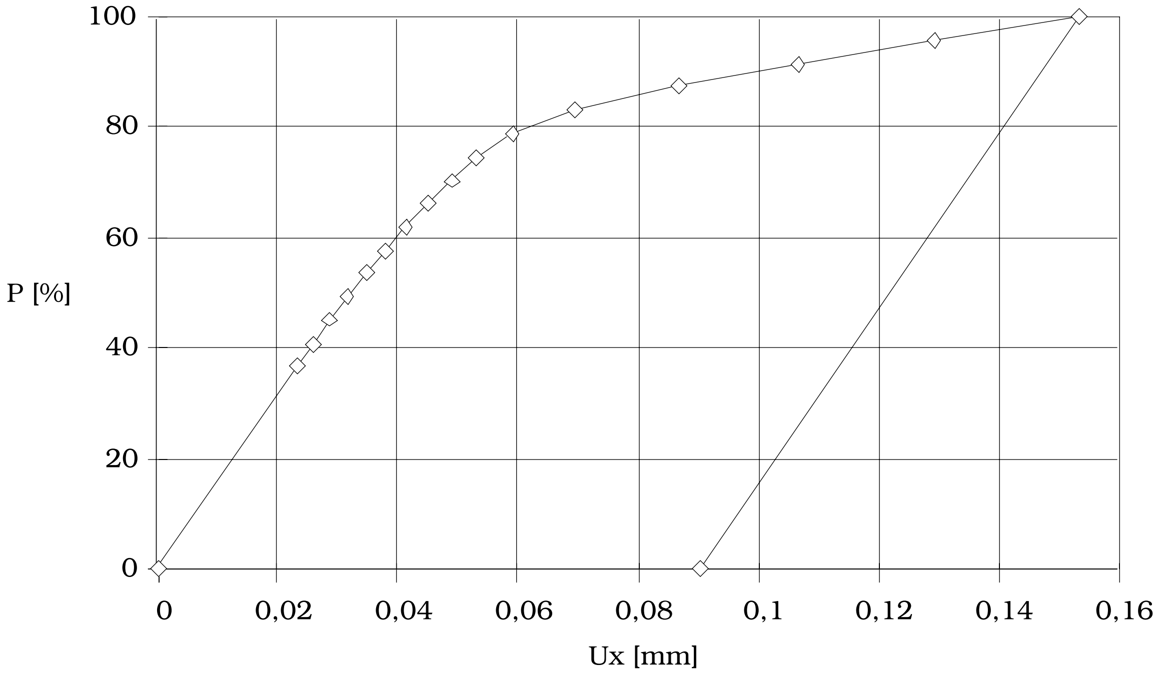


Fig. 3.11: Load-Deflection diagram of perforated plate (point D)

The diagram shows that a plastic deflection remains after the load has been detached.
The results of the calculation, concerning the development of the yield zone and the displacements, conform with the results of Zienkiewicz, Valliapan and King/10/ for the same problem.

PlastFEM Shareware Info

The registrated version of this document contains the missing theory chapters 4 to 6 (35 pages). For further details, please see the *ReadMe* file and the *Register* program that come with the shareware distribution of *PlastFEM*.

7. References

/1/ Bathe, K.-J.:
Finite Element Procedures in Engineering Analysis; Prentice Hall (1982)

/2/ Benham, P.P., Crawford, R.J., Armstrong, C.G.:
Mechanics of Engineering Materials; 2nd edition, Addison-Wesley Pub Co. (1996)

/3/ Kessel, S., Fröhling, D.:
Technische Mechanik/Technical Mechanics; bilingual textbook, B.G. Teubner Stuttgart Leipzig (1998)

/4/ Meek, J.L.:
Computer Methods in Structural Analysis; E & FN Spon London New York Tokyo Melbourne Madras (1991)

/5/ Nayak G.C., Zienkiewicz, O.C.:
Note on the 'Alpha'-Constant Stiffness Method for the Analysis of Non-Linear Problems; Int. J. num. Meth. Engng., Vol. 4, S. 579-582 (1972)

/6/ Nayak G.C., Zienkiewicz, O.C.:
Elasto-Plastic Stress Analysis. A Generalisation for Various Constitutive Relations Including Strain Softening; Int. J. num. Meth. Engng., Vol. 5, S. 113-135 (1972)

/7/ Sadek, E.A.:
A Scheme for the Automatic Generation of Triangular Finite Elements; Int. J. num. Meth. Engng., Vol. 15, S. 1813-1822 (1980)

/8/ Yamada, Y., Yoshimura, N., Sakurai, T.:
Plastic Stress-Strain Matrix and Its Application for the Solution of Elastic-Plastic Problems by the Finite Element Method; Int. J. Mech. Sci., Vol. 10, S. 343-354 (1968)

/9/ Zienkiewicz, O.C.:
Methode der finiten Elemente; Carl Hanser Verlag München, 2. Auflage, 1984

/10/ Zienkiewicz O.C., Valliappan, S. King, I.P.:
Elasto-Plastic Solutions of Engineering Problems, 'Initial Stress', Finite Element Approach; Int. J. num. Meth. Engng., Vol. 1, S. 75-100 (1969)