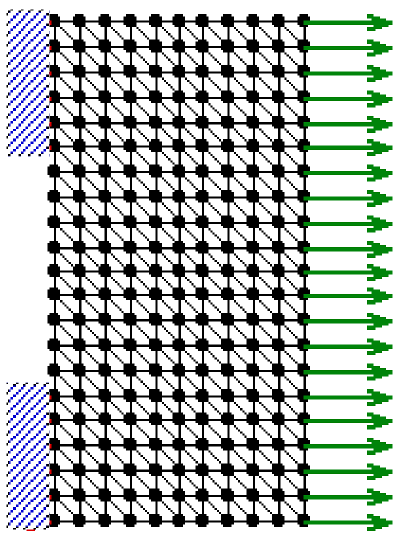
Module I:

Two-dimensional linear elasticity

• application notes and tutorial

• Problems



selected excerpts from Read Me file for:

ElemFin 1.1.1

Yannick CALLAUD in Symantec C++.
1 place of Falleron, 44300 NANTES FRANCE, Email: yc16@calva.net

1/ Introduction :

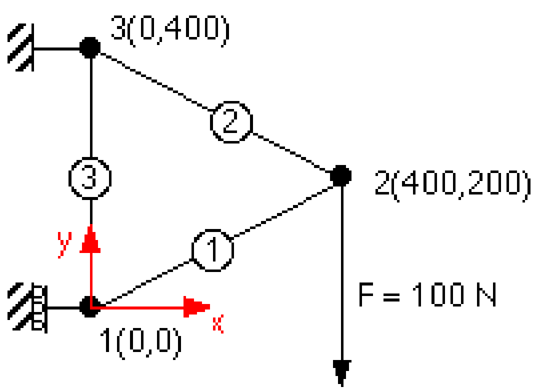
ElemFin is a calculation programme of the structures based on the finite element method.
(It appears based on the various assumptions of linear elasticity).

It makes it possible to solve plane problems with elements:
- bars.
- beams.
- triangulated plates.

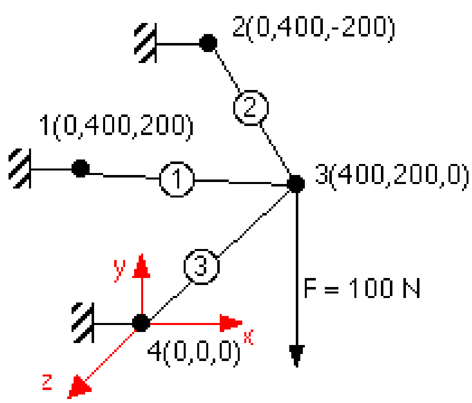
ElemFin allows the calculation of:
- displacements of each nodes.
- reactions to the supports.
- stress in each element.

2/ Some provided examples :

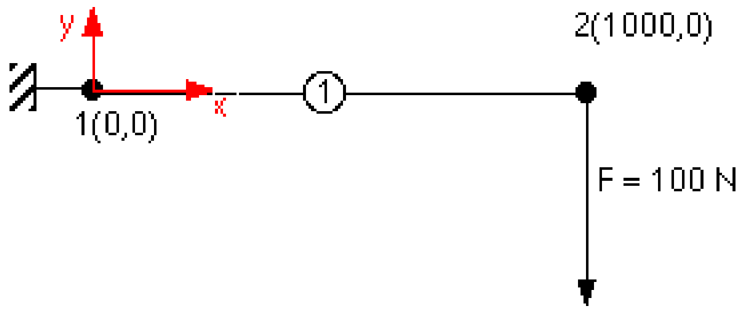
a / **Bars 2D** : *(Recall Castigliano's Theorem)*



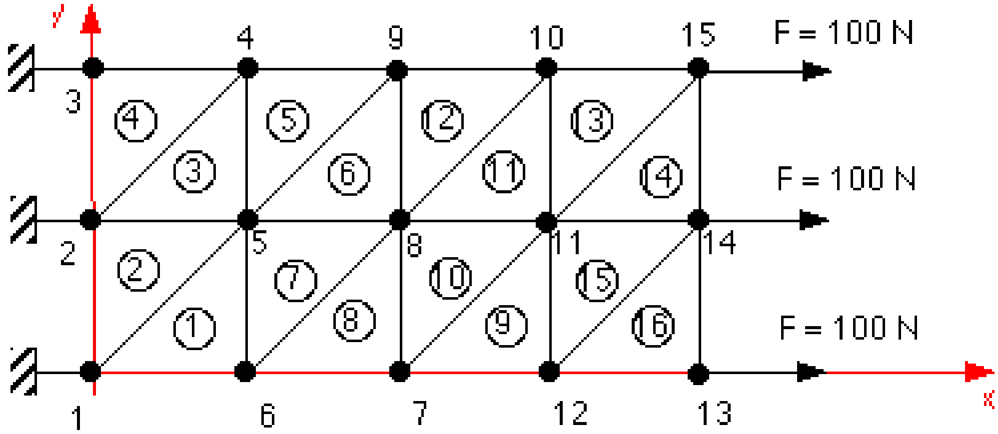
b / **Bars 3D** :



c / **Beams 2D** :



d / **Stresses plane** : with fully constrained nodes 1,2,3

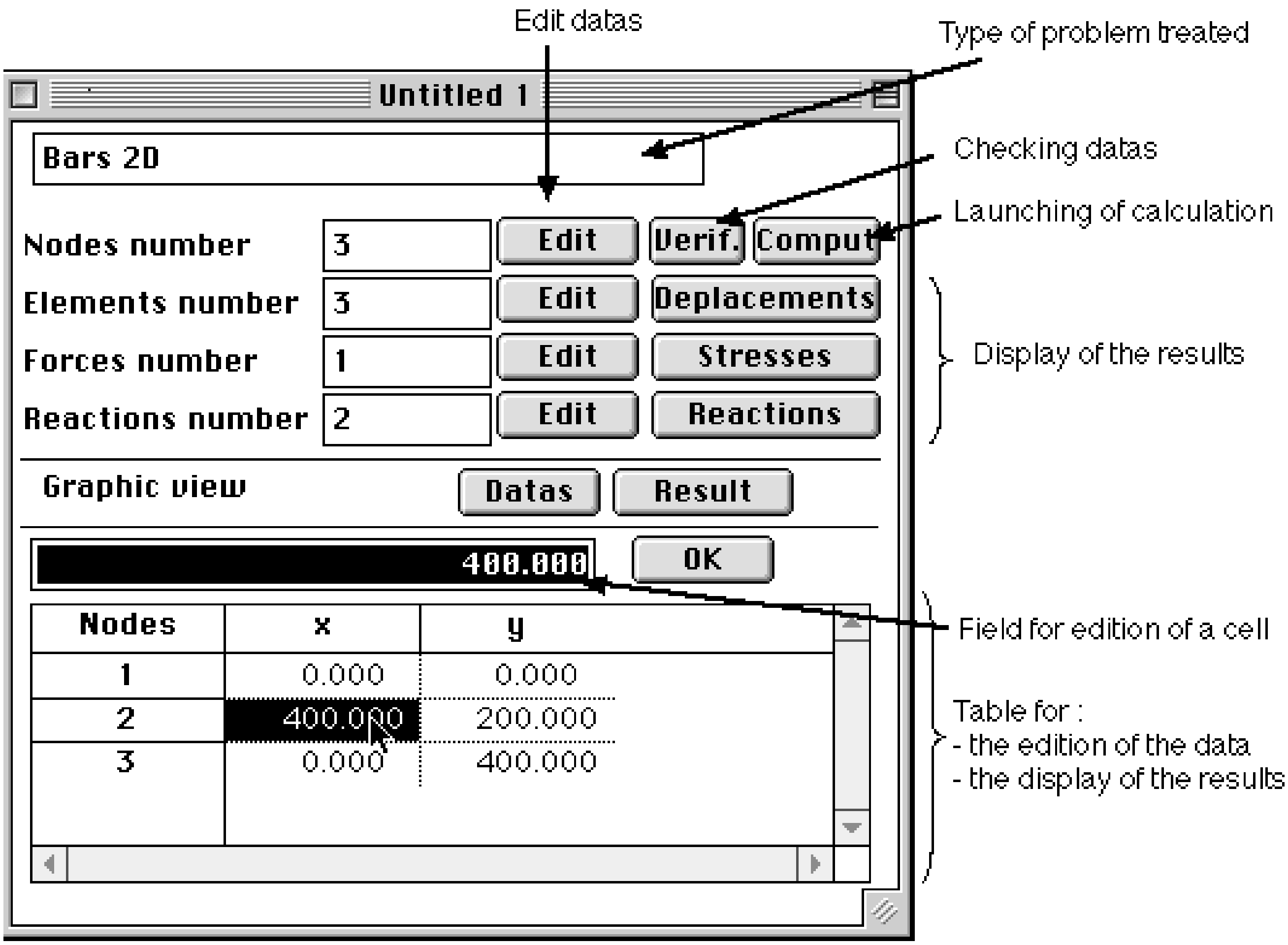


Note: the elements must be described by indicating the numbers of the nodes
in the trigonometrical direction (direction reverses with the needles of a watch).

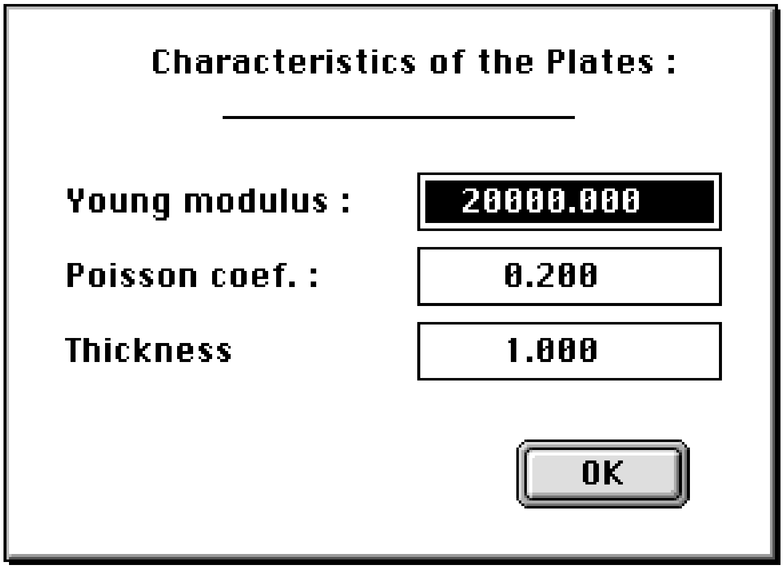
e / **Plane Deformations** :
(similar to plane stresses)

3/ **Screens** :

a / **The principal screen** :



e / **Characteristics of the Plates** :

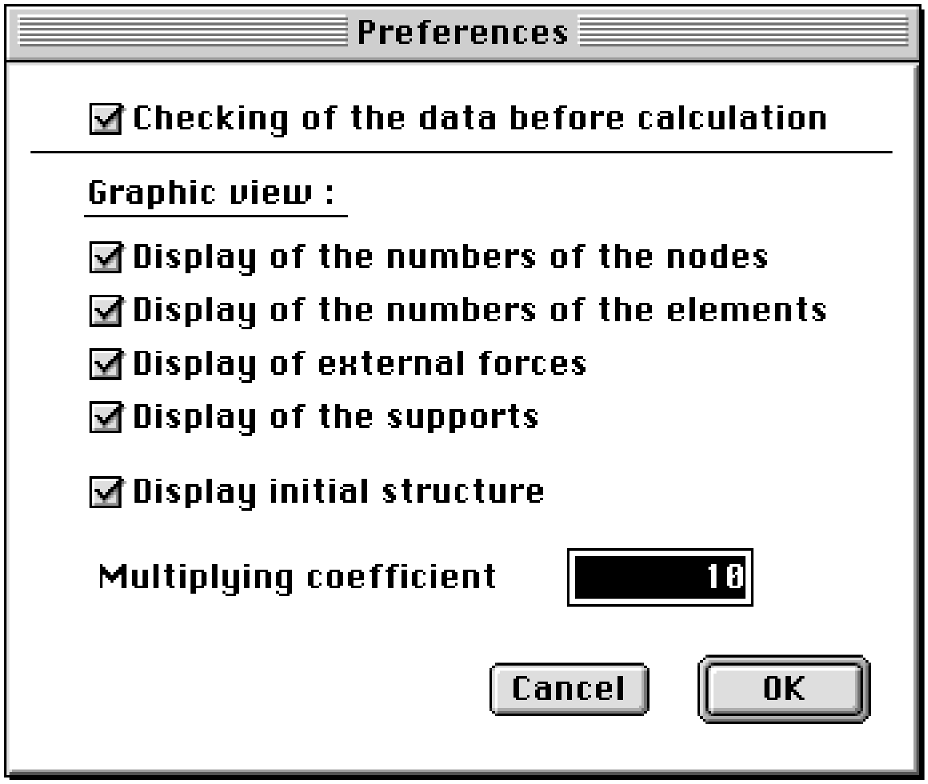


Young modulus: longitudinal modulus of elasticity of material employed, it is often called ' E'.

Poisson coefficient : negative ratio of transverse and longitudinal strains $\nu = -\varepsilon_y / \varepsilon_x$.

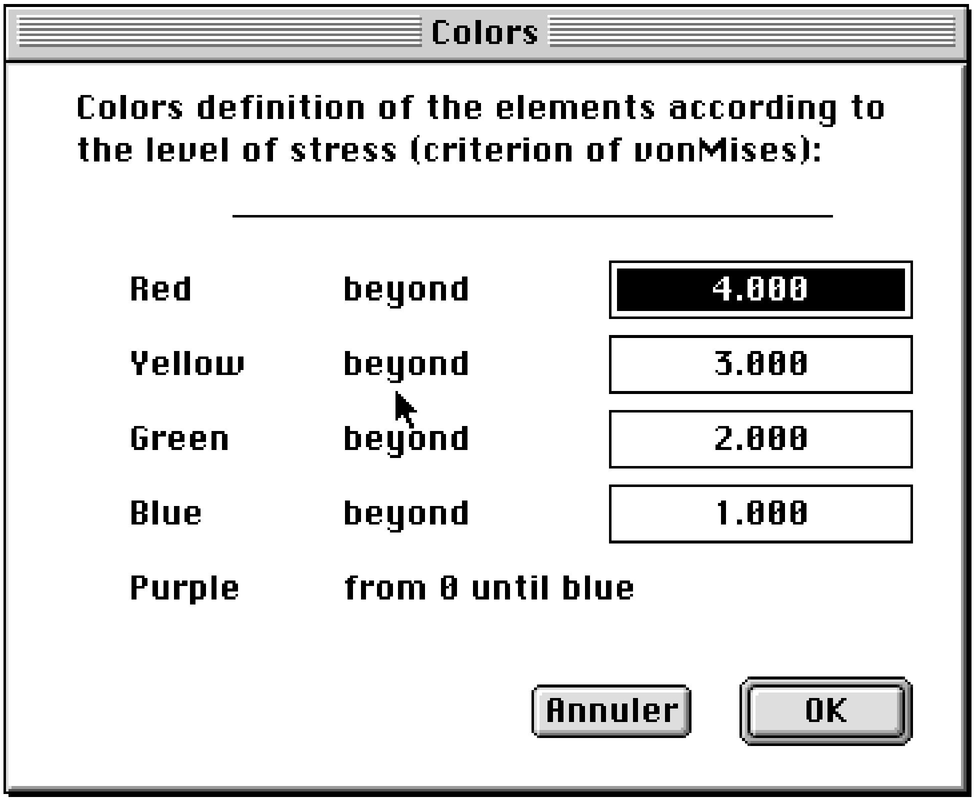
Thickness: thickness of the plate or the slice considered.

f / **Preferences** :



Checking of the data before calculation: The program checks the coherence of the data before undertaking calculation. Examples might include inconsistent mesh numbering etc.
Display of the numbers of the nodes: the numbers of the nodes are showed near the nodes of the structure.
Display of the numbers of the elements: the numbers of the elements are showed near the bars or in the triangles representatives the elements of the structure.
Display of external forces: the forces are represented by vectors.
Display of the supports: the supports are symbolized with the nodes where they are applied.
Display initial structure: The structure in initial position is represented in grayed during the graphic display of the results.
Multiplying coefficient: To represent the deformed structure it is necessary to amplify the deformations so that they are visible and not infinitesimal.

g / **The codes colors** :



The elements are drawn in the color corresponding to the level of the internal stresses. The program calculation the vonMises (maximum principal) stress and then selected the adequate color. In this example:

- the purple elements have an internal stress ranging between 0 and 1.
- the blue elements have an internal stress ranging between 1.001 and 2.
- the green elements have an internal stress ranging between 2.001 and 3.
- the yellow elements have an internal stress ranging between 3.001 and 4.
- the red elements have an internal stress higher than 4

4 / **Characteristics of some materials**:

E.10^9 Pa: Young N or modulus of elasticity υ : numbers of Poisson

σ1 . 10^6 Pa: elastic limit in traction ρ . 10^3 kg/m^3: density

Material	E	v	σ1	ρ
Iron	200,0	0,24	200	7,80
Steel 45SCD6	220,0	0,28	1450	7,80
Stainless steel 18.10	203,0	0,29	200	7,90
Current gray cast iron	90,0	0,29	190	7,20
Titane	105,5	0,34	300	4,50
Alloy titanium TA 6 V	105,0	0,34	900	4,42
Aluminium	70,5	0,34	150	2,70
Alloy AU 4 G	75,0	0,33	200	2,80
Alloy AU 2 GN	75,0	0,34	370	2,80
Zicral AZ 8 GU	72,0	0,34	550	2,80
Copper	100,0	0,33	180	8,90
Brass	92,0	0,33	200	7,30
Bronzes ordinary	106,0	0,31	240	8,40
Bronze with beryllium	130,0	0,34	800	8,25
Beryllium	300,0	0,05	300	1,85
Magnésium	46,0	0,34	180	1,74
Zinc	130,0	0,21	120	7,15
Nickel	205,0	0,31	300	8,90
Plexiglass	2,9	0,40	80	1,80
Glass	60,0	0,24	60	2,50
Araldite	3,0	0,40	70	1,15

5 / **History of the programs:**

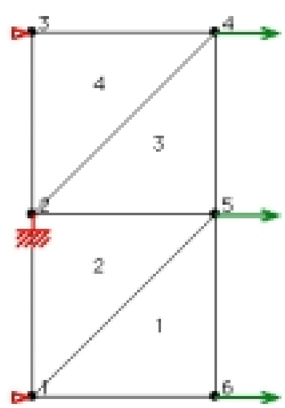
- version 1.0.0: First version placed at the disposal of the public.
- version 1.0.1:
  - . Addition validation and jump with the following cell with the keys, return and enter at the time of the edition of a cell.
  - . Addition of the on line help thanks to the bubbles of assistance.
  - . Correction of a bug during the graphic display of the result for the bars.
- version 1.1.0:
  - . The titles of the columns and the lines are now fixed when one scrolle.
  - . All comments are now in the resources of the program.
  - . Addition of the von Mises stress.
  - . Addition of new preferences "codes colors ". These preferences are stored in the document.
  - . Display color of the elements according to the internal stress vonMises.
  - . The files are not compatible any more with the old version of ElemFin, on the other hand the old files can be opened by this new version.
  - . Addition of a file ElFin prefs in the file preference of the file sytem.
- version 1.1.1:
  - . Correction of a bug which prevented the application to leave correctly.
  - . Vérification of the memory size before allocating the matrices for calculation.
  - . Better management of column titles.

PROBLEM 1 - Tension Test Checks of ElemFin 1.1.1

PURPOSE: It is first of all important to examine the accuracy of a numerical code by setting up a simple numerical experiment, which in this linear elasticity module, is just a tension test of a plate of material. This will be done both with a small mesh of your own creation and then a larger mesh provided to you. In both cases, you need to set up suitable boundary conditions. In the second problem of this module, you will adjust boundary conditions in the tension test in order to simulate single cell deformation when it is partially attached to a rigid substrate and then pulled upon. This latter problem aims to reveal a relation between the Young's modulus of a cell and both how the cell is attached and how it deforms.

A. You should first set up your own 6-node, 4-element mesh (as below). Click on the "Edit" button for "Nodes number" and put node 1 at the origin, clicking OK after each x and y entry in the databox. Make the length between nodes 1 and 2 equal to 50 (units of nodal force/E). Be sure to also create or "Edit" the element numbers. If you have any questions, first look at the 400 element mesh. Specify suitable boundary conditions for the tension test below. The soundest approach is to: (1) fully fix one support (node 2) among the supports constrained to slide only in the y-direction (nodes 1 and 3), and (2) use half-forces at the end nodes (nodes 4 and 6 below) and full forces at non-corner nodes (eg. node 5). The initial lengths of the plate in the x and y directions are L_x^o, L_y^o, respectively.

QUESTION: In two brief sentences, explain why these two suggestions make sense.



Specify plate characteristics for E and ν of plexiglass, and complete the following:

E = __2900_____ ν = __________ thickness = ___1______

L_x^o = __________ L_y^o = __________

Applying suitable nodal forces in the x-direction (F_x = applied σ_x * thickness * L_y^o) which achieve the applied stresses in the following table, compute the resulting deformation and stresses and complete the table. In examining the stress field (stresses are always in units of E, i.e. 10^6 Pa), one should find uniform σ_x and vanishing $\sigma_y = \sigma_{xy} = 0$; check that this is the case. Fill in the last column, E_{fit}, using the standard definition for E = stress / strain.

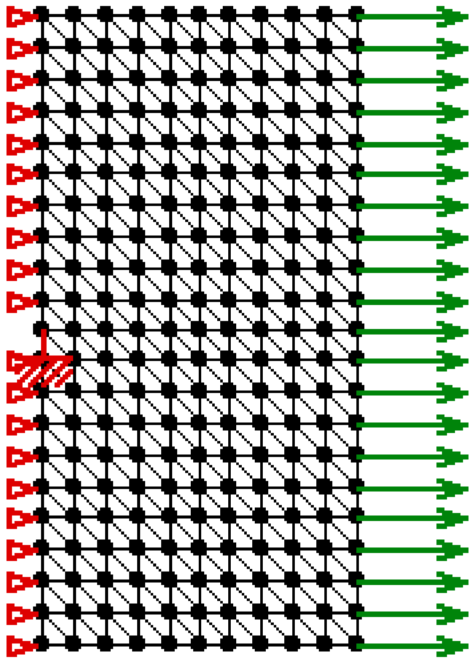
applied σ_x	L_x	L_y	uniform* ?	ε_x	ε_y	$-\varepsilon_y/\varepsilon_x$	E_{fit}
0							------------
10							
100							

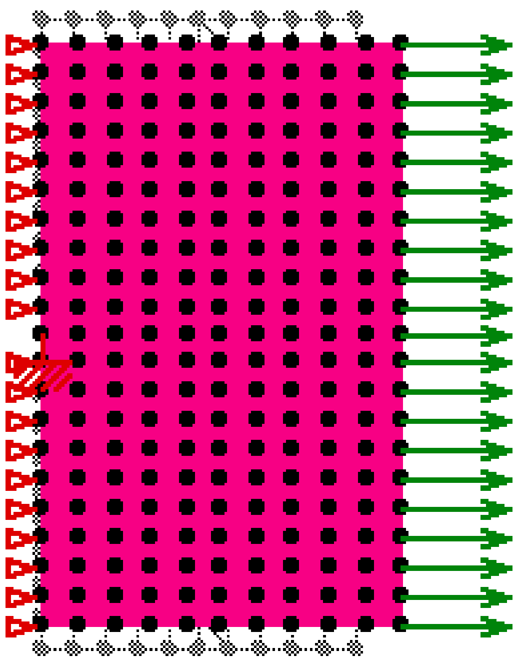
*The question is whether the stresses in the plate and strain appear uniform.

QUESTION: Is the deformation a plane strain or a plane stress deformation?

QUESTION: How does E_{fit} compare to the specified E=2900?

B. Now use the given 400 element mesh with suitable boundary conditions. Use the same E and ν for plexiglass, and again complete the table below.





E = ____________ ν = ___________ thickness = ___________

L_x^o = ___________ L_y^o = ___________

applied σ_x	L_x	L_y	uniform* ?	ε_x	ε_y	$-\varepsilon_y/\varepsilon_x$	E_{fit}
0							------------
100							

QUESTION: How do the two tension test computations compare - similar or different?