ElemFin 1.2.2

1/ Introduction :

ElemFin is a calculation programme of the structures based on the principle of the finite element method.

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

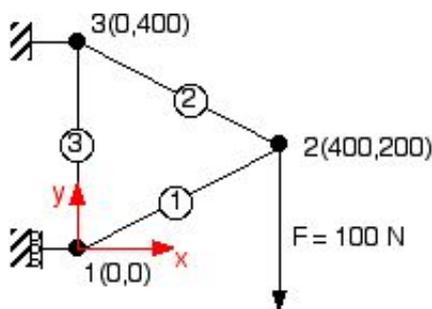and of the space problems with elements:
- bars.
- beams.

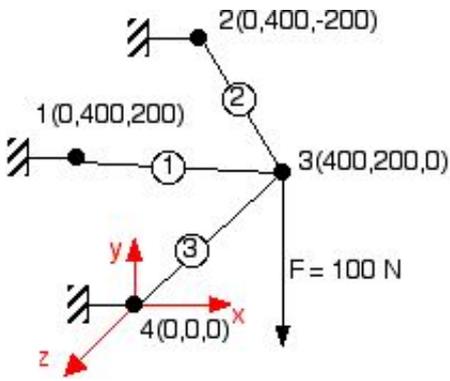ElemFin allows the calculation of:
- displacements of each nodes.
- reactions to the supports.
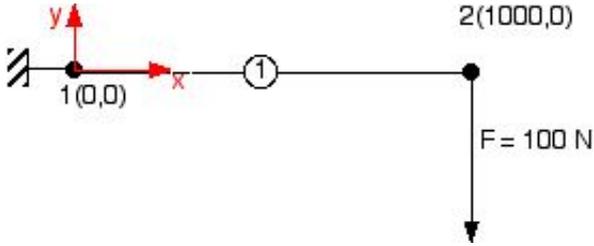- stress in each element.

2/ Some provided examples :
a / Bars 2D :

b / Bars 3D :
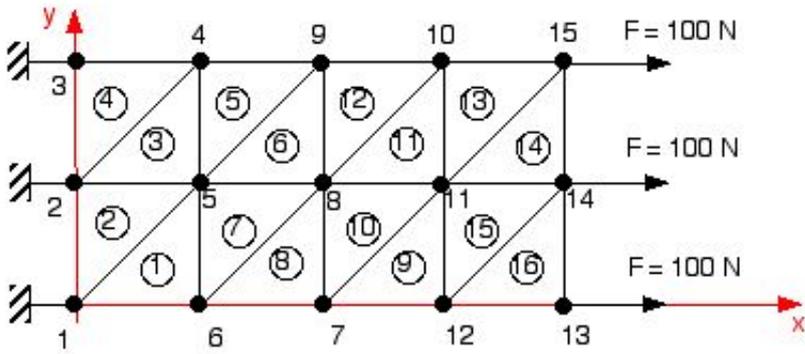
2(0,400,-200)

1(0,400,200)

3(400,200,0)

F = 100 N

4(0,0,0)

c / Beams 2D :

2(1000,0)

1(0,0)

F = 100 N

d / Stresses plane :

F = 100 N

F = 100 N

F = 100 N

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 :
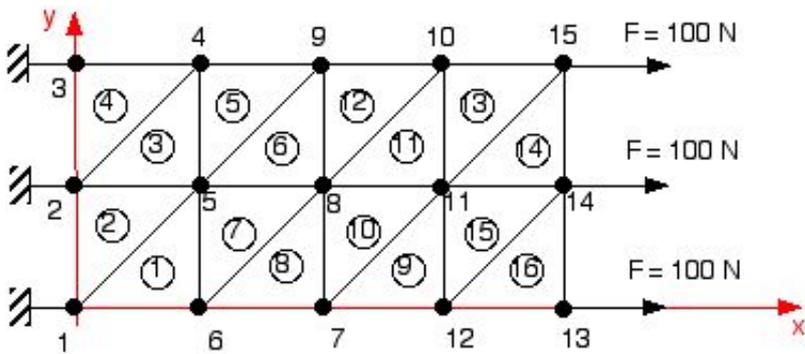

F = 100 N

F = 100 N

F = 100 N

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).

3/ <u>Screens</u> :
a / The principal screen :

Edit datas

Type of problem treated

Checking datas

Launching of calculation

Untitled 1

Bars 2D

Nodes number	3	Edit	Verif. Comput
Elements number	3	Edit	Deplacements
Forces number	1	Edit	Stresses
Reactions number	2	Edit	Reactions

Display of the results

Graphic view   Datas   Result

400.000   OK

Field for edition of a cell

Nodes	x	y
1	0.000	0.000
2	400.000	200.000
3	0.000	400.000

Table for :
- the edition of the data
- the display of the results

b / Characteristic of the Bars 2D and 3D :

Untitled 1

Bars 2D

Nodes number	3	Edit	Verif. Comput
Elements number	3	Edit	Deplacements
Forces number	1	Edit	Stresses
Reactions number	2	Edit	Reactions

Graphic view   Datas   Result

OK

Elements	Node 1	Node 2	Young	Section
1	1	2	20000	12.000
2	2	3	20000	12.000
3	3	1	20000	12.000

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

<u>Section</u> : The sectional area of the beam.

c / Characteristic of the Beams 2D :
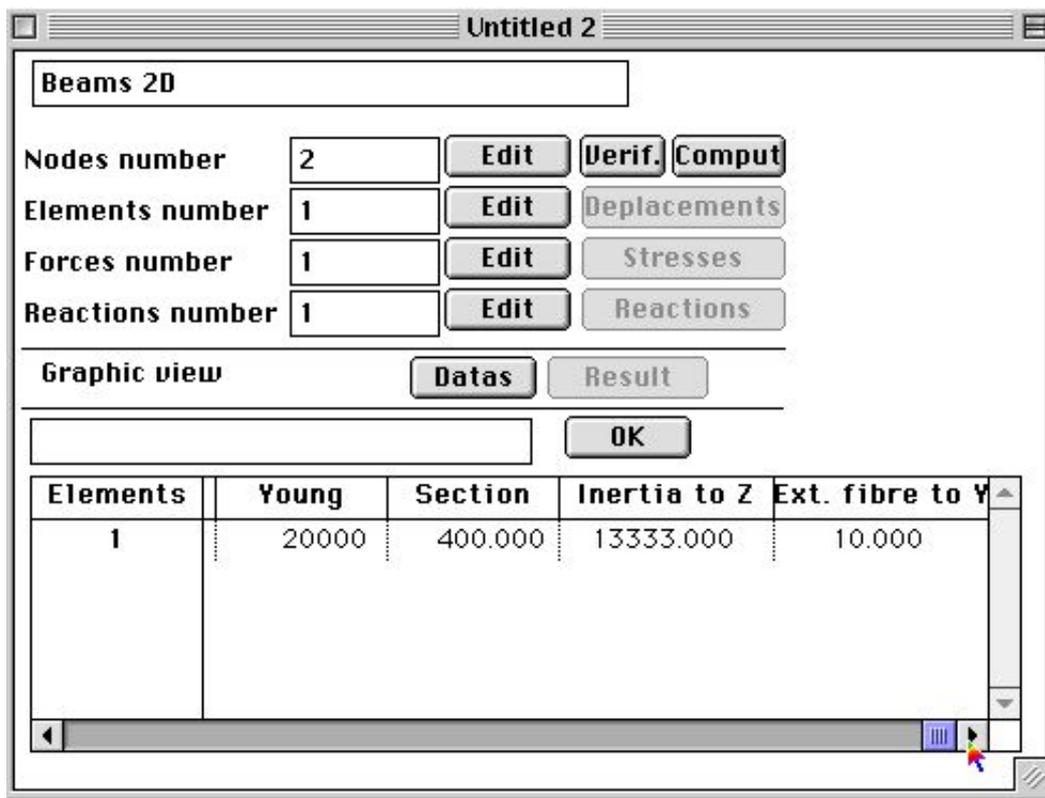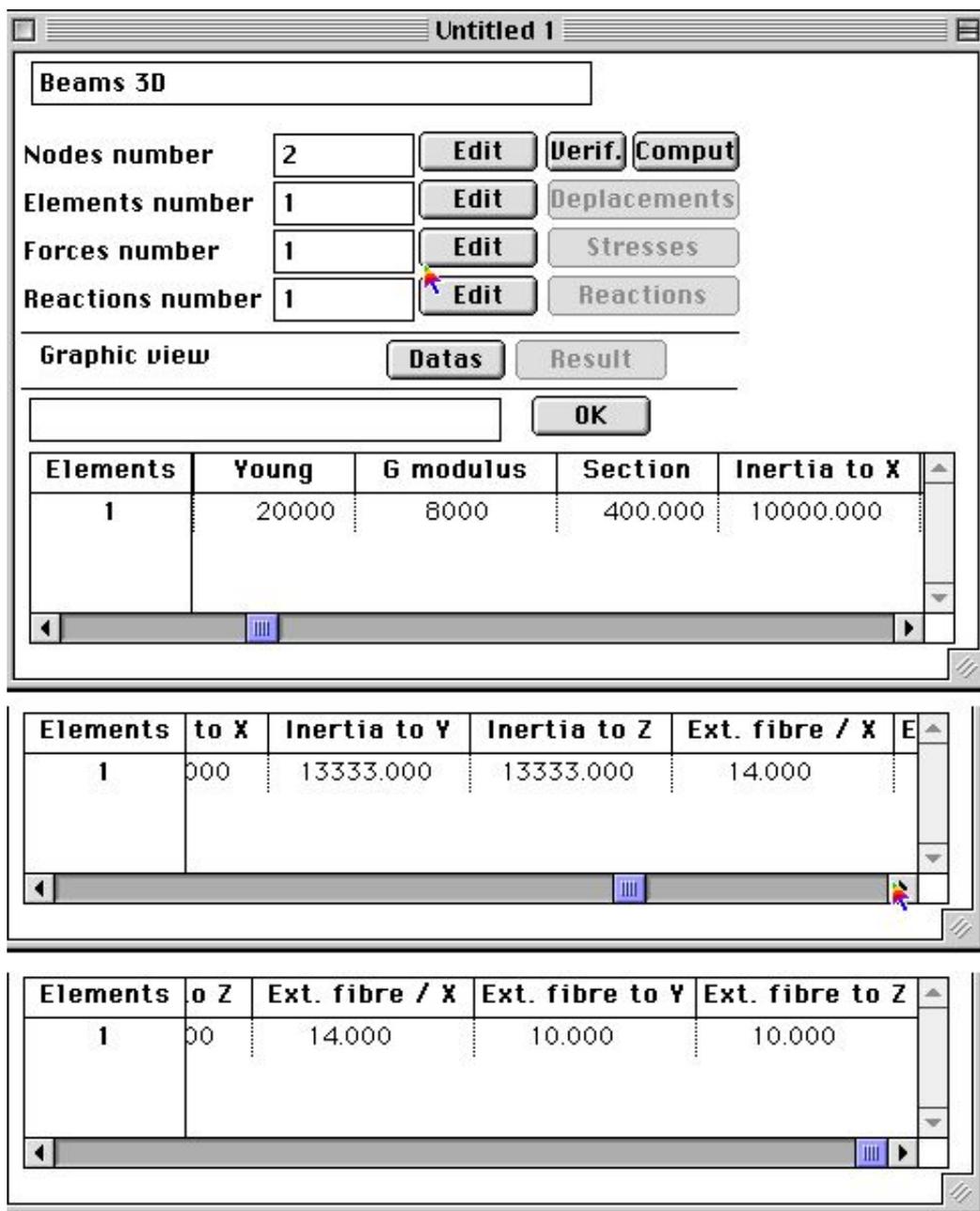

<u>Young modulus</u> : longitudinal modulus of elasticity of material employed, it is often called ' E'.

<u>Section</u> : The sectional area of the beam.

<u>Inertia according to Z</u> : or quadratic Moment $I_Z = S \, D_S \cdot x^2$.

<u>Outdistance between external fibre and neutral fibre</u> : this distance is used for calculation the stresses.

d / Characteristic of Beams 3D :

Beams 3D

Nodes number	2	Edit Verif. Comput
Elements number	1	Edit Deplacements
Forces number	1	Edit Stresses
Reactions number	1	Edit Reactions

Graphic view   Datas   Result

OK

Elements	Young	G modulus	Section	Inertia to X
1	20000	8000	400.000	10000.000

Elements	to X	Inertia to Y	Inertia to Z	Ext. fibre / X	E
1	000	13333.000	13333.000	14.000	

Elements	o Z	Ext. fibre / X	Ext. fibre to Y	Ext. fibre to Z
1	00	14.000	10.000	10.000

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

<u>G modulus (trans.)</u>: transverse modulus of elasticity or module of Coulomb.

<u>Section</u>: The sectional area of the beam.

<u>Inertia according to X</u>: or quadratic Moment $I_x = S\ D_s.\ y^2$.

<u>Inertia following Y</u>: or quadratic Moment $I_y = S\ D_s.\ z^2$.

<u>Inertia according to Z</u>: or quadratic Moment $I_z = S\ D_s.\ x^{2.}$

<u>Distances between external fibres and neutral fibre</u>: these distances are used for calculation of stresses.

e / Characteristics of the Plates :

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

Poisson coef. : coefficient which connects dilations longitudinal and transverse ey = - n.ex.

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.

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.
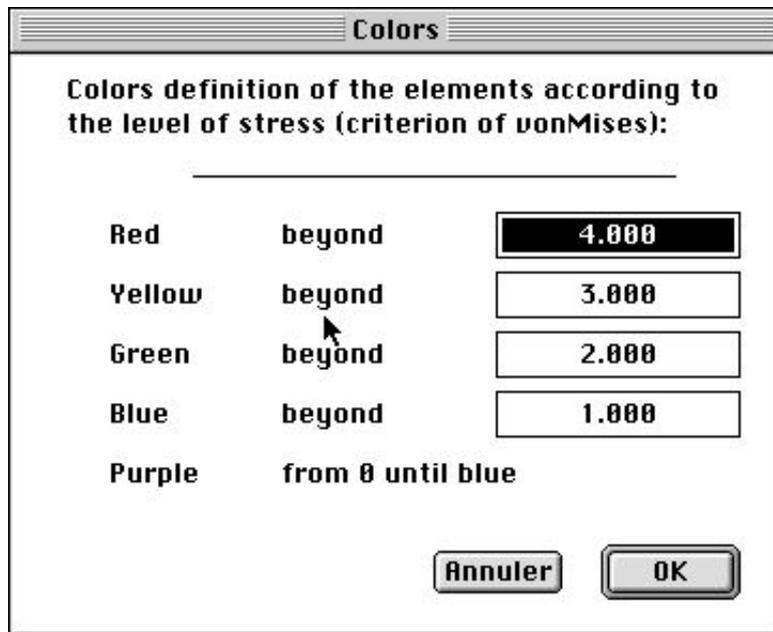
Display of external forces: the forces are represented by vectors.

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.

<u>Multiplying coefficient</u>: To represent the deformed structure it is necessary to amplify the deformations so that they are visible .

g / The codes colors :

```
═══════════════ Colors ═══════════════

   Colors definition of the elements according to
   the level of stress (criterion of vonMises):
   _____

      Red        beyond          ┌──────────┐
                                  │  4.000   │
                                  └──────────┘
      Yellow     beyond          ┌──────────┐
                                  │  3.000   │
                                  └──────────┘
      Green      beyond          ┌──────────┐
                                  │  2.000   │
                                  └──────────┘
      Blue       beyond          ┌──────────┐
                                  │  1.000   │
                                  └──────────┘
      Purple     from 0 until blue

                            ┌─────────┐  ┌─────────┐
                            │ Annuler │  │   OK    │
                            └─────────┘  └─────────┘
```

The elements are drawn in the color corresponding to the level of the internal stresses. The program calculation the vonMises 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 / <u>Characteristics of some materials</u>:
$E.10^9$Pa: Young N or modulus of elasticity
u : numbers of Poisson
$s1 : 10^6$Pa: elastic limit in traction
$r : 10^3$ kg/m^3: density

Material	E	n	s1	r
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 / Evolutions 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.

•version 1.1.2 :

. Add the possibilité to edit a group of cells with commands (Copy, Cut,

Paste, Delete.)

. It's possible to move into table with arrow keys.
. Correct gestion of printing.
•version 1.2.0 :
. It's now possible to attribute individual characteristic for each element bar
or beam.
. The error messages mention the line numbers with problem.
. Checking if some elements are setting with same nodes.
. The error messages are in resource: STR 1000.
. Correction of a bug during Copy Paste.
. The stresses bars are now positive if extention or negative if compression.
. Correct error in rigidity matrix of elem beam 3D.
. Correct calculation of beams 3D deformation.
. Modification of printing.
•version 1.2.1 :
. Correct error if calculation without datas checking.
. A handle was not free when you closed a window.
•version 1.2.2 :
. Correct error in calculation of Beams 3D stresses.

6 / **the author** :

Program writes by Yannick CALLAUD in Symantec C++.

7/ **Errors :**

My English is not good. If you find an error, please send to me an email with the correction.
Thank you.

<retour> <Home Page>