Structurix

0/ Utilisation :

Structurix is a freeware. It is a gratis application, usable for as long as you wish; you may copy and distribute it at no charge.

You expressly acknowledge and agree that use of the Structurix software is your own risk.

Structurix is distributed "as-is" and without warranty, express or implied. I am not in any way responsible for software or hardware damage that could result from its use.

1/ Introduction :

Structurix is a calculation program: the analysis of structures based on the principle of the finite element method.

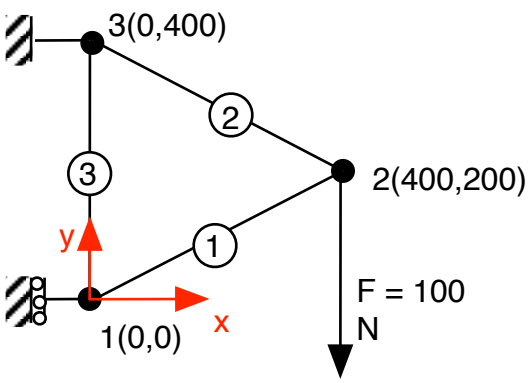
It makes it possible to solve plane problems with elements:
- bars.
- beams.
- triangulars.

and of the space problems with elements:
- bars.
- beams.
- rectangular element for the calculation of the board flex
- triangular element for the calculation of revolution solid

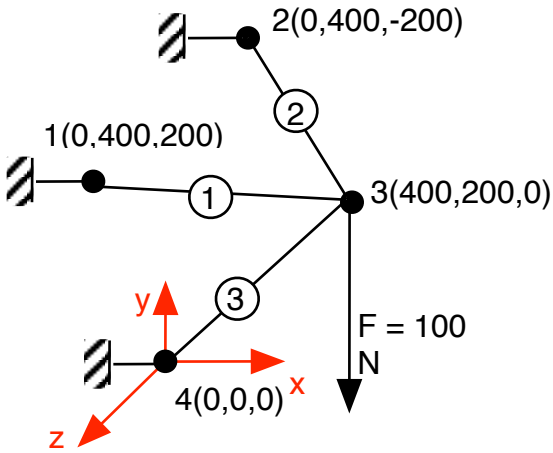
ElemFin allows the calculation of:
- displacements of each nodes.
- reactions to the supports.
- stress in each element.

2/ Some provided examples :

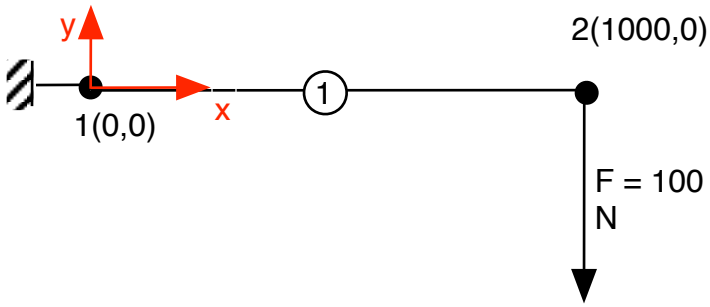
2.1 / Bars 2D :



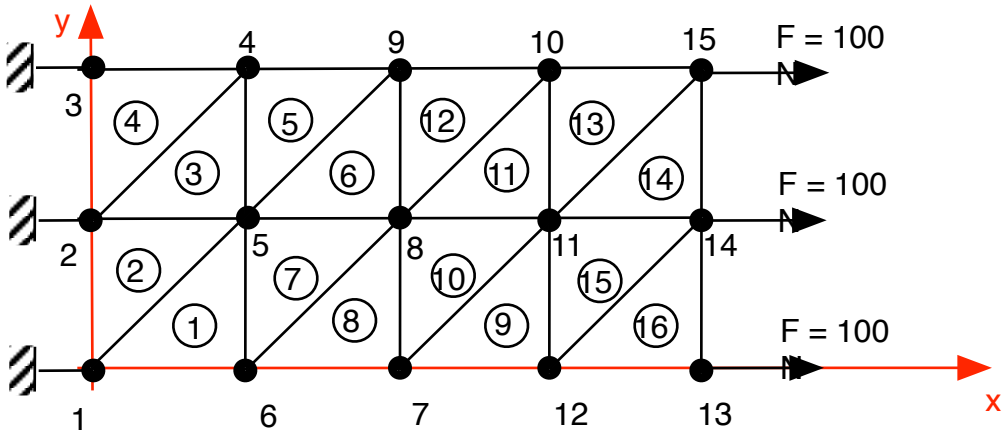
2.2 / Bars 3D :



2.3 / Beams 2D :



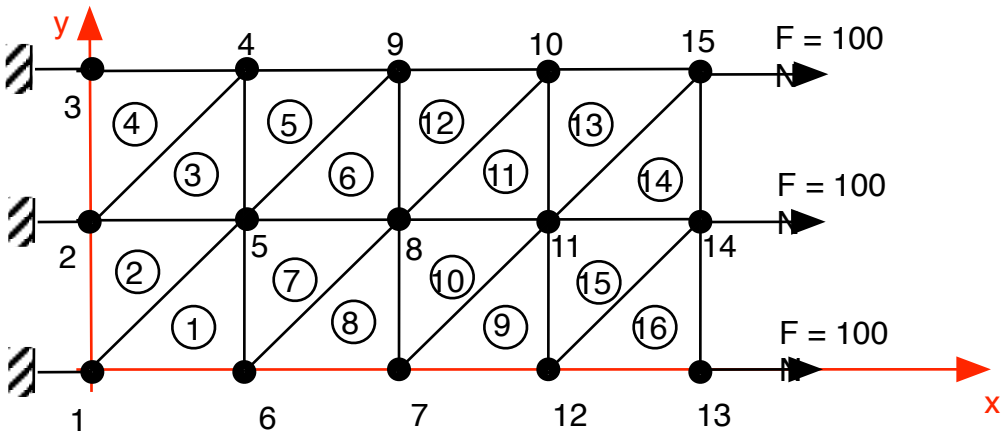
2.4 / Stresses plane :



The model “stresses plane” is used for piece with constant thickness which represents a cutout in a sheet of metal. For exemple: fork wrench, neck swan...

Note: the elements must be described by indicating the numbers of the nodes in the `counterclockwise direction`.

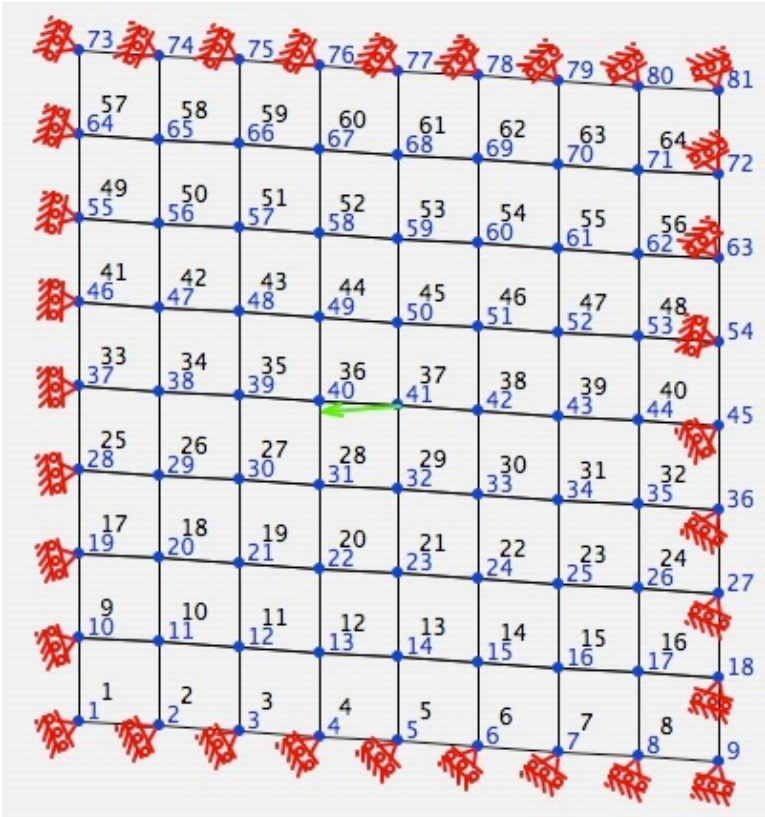
2.5 / Plane Deformations :



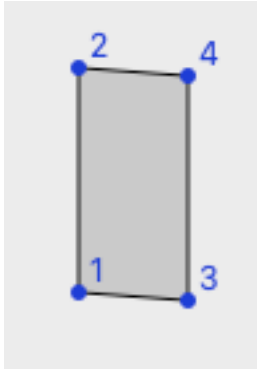
The model “plane deformations” is used for slice of structure where there is no deformation on z direction considered.
For example: a section of a dam...

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

2.6 / Board flexion :



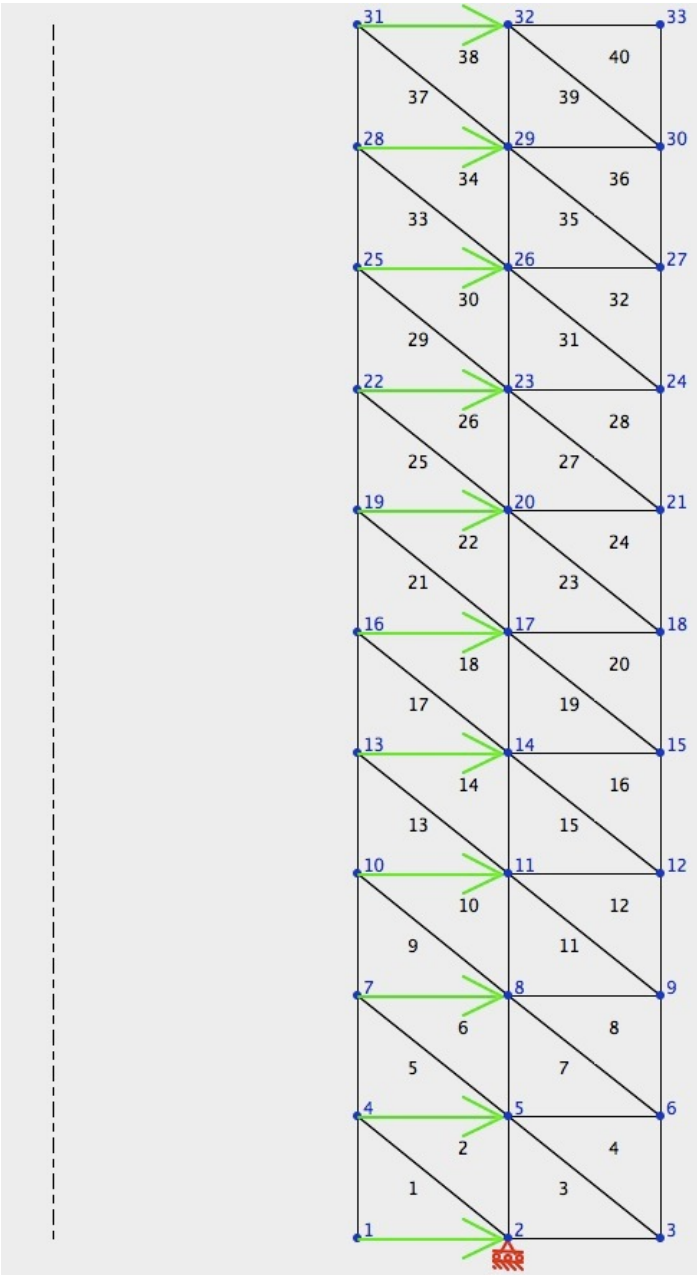
The model “Board flexion” is used for board with small thickness, witch perpendicular effort, and small deformations regarding to thickness.
For exemple: Floor



Note: the elements must be described by indicating the numbers of the nodes in the correct direction.

the first node : must be the bottom left.
the second : must be the top left
the third : must be the bottom right
the fourth : must be the top right.

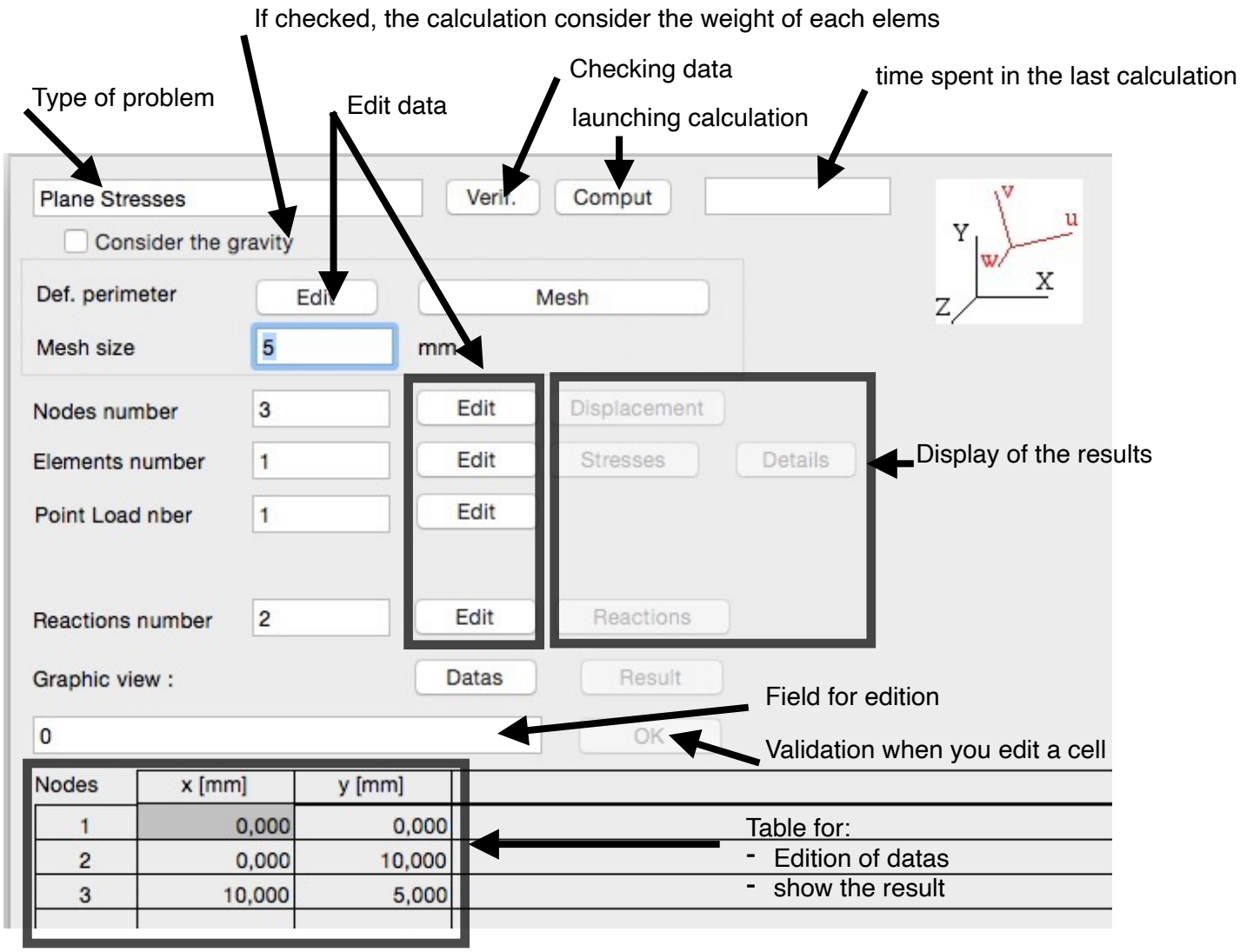
2.7/ Revolution solid :



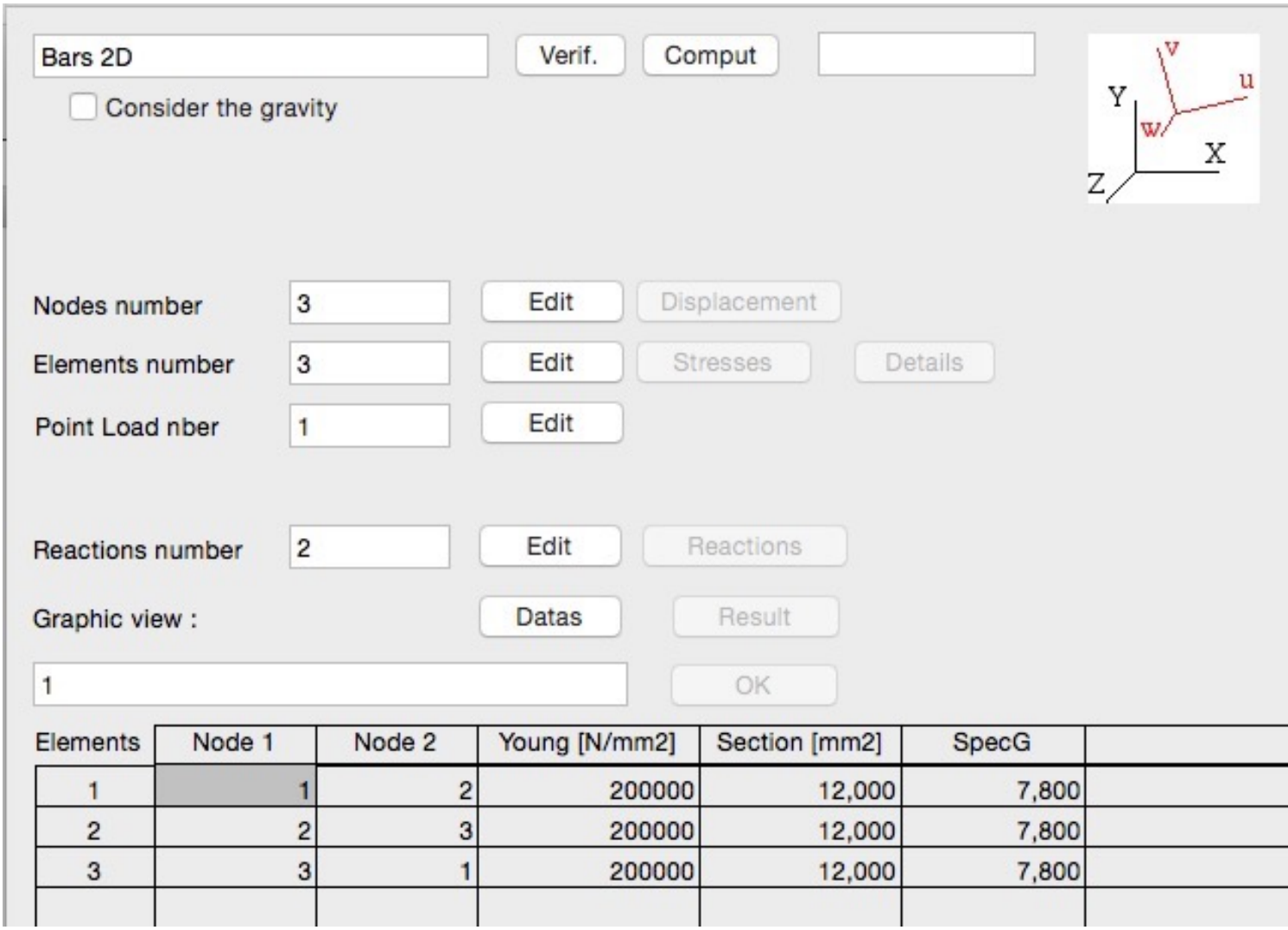
The model “Solid of Revolution” is used for volumetric parts. Volume must be generated by a surface that rotates about an axis.
For exemple: Tube, cylinder, punch.
Note: the elements must be described by indicating the numbers of the nodes in the counterclockwise direction.

3/ Screens :

3.1 / The principal screen :



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



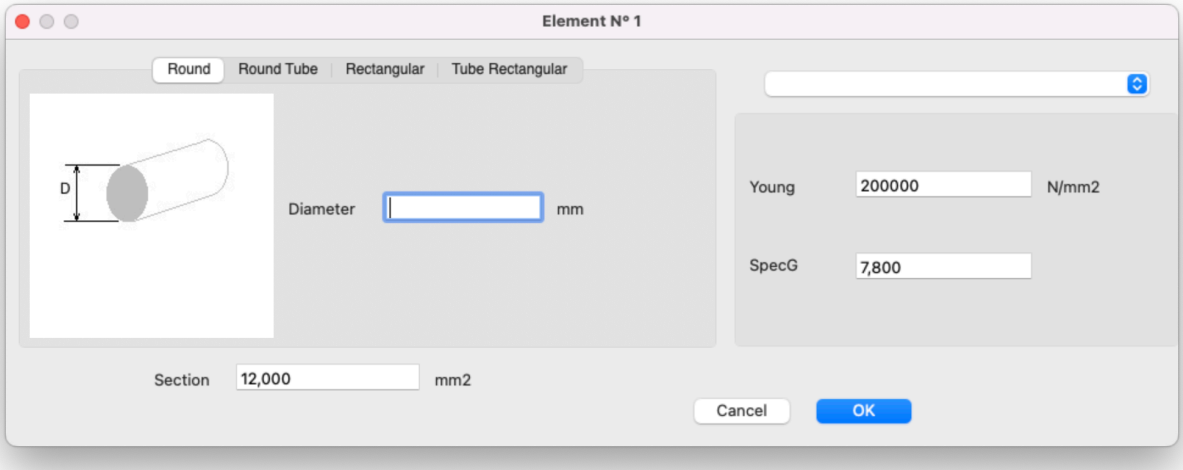
Elements	Node 1	Node 2	Young [N/mm2]	Section [mm2]	SpecG	
1	1	2	200000	12,000	7,800	
2	2	3	200000	12,000	7,800	
3	3	1	200000	12,000	7,800	

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

Section : The sectional area of the beam.

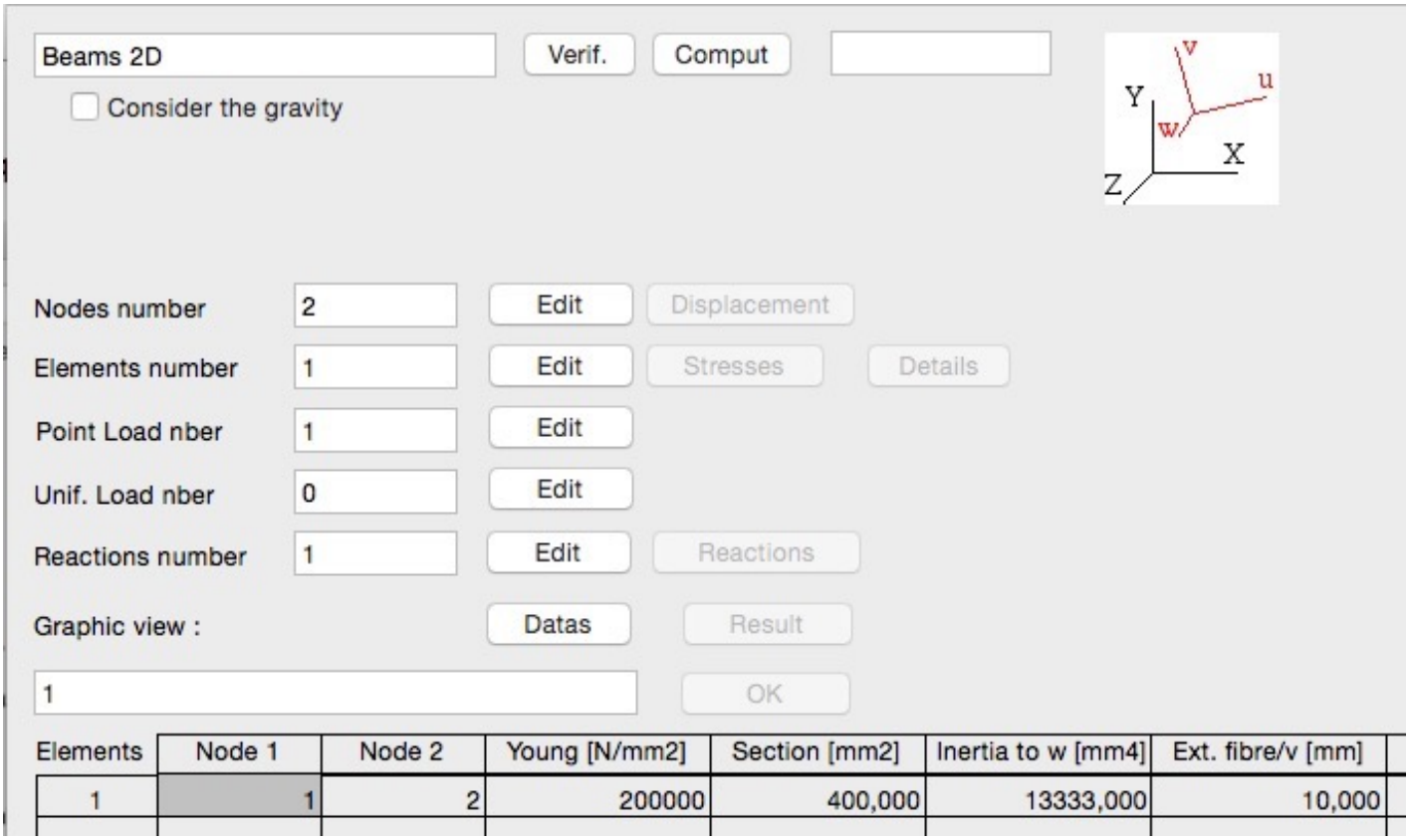
SpecG : The specific gravity of material used. It's ratio between masse for a volume of material and the mass for a same volume of water.

A double click on a line, opens an editing window for the item.



This window lets you choose the shape of the beam and its material.
You can also enter the desired values if necessary.

3.3 / Characteristic of the Beams 2D :



Elements	Node 1	Node 2	Young [N/mm2]	Section [mm2]	Inertia to w [mm4]	Ext. fibre/v [mm]
1	1	2	200000	400,000	13333,000	10,000

Elements	e 1	Node 2	Young [N/mm2]	Section [mm2]	Inertia to w [mm4]	Ext. fibre/v [mm]	SpecG
1	1	2	200000	400,000	13333,000	10,000	7,800

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

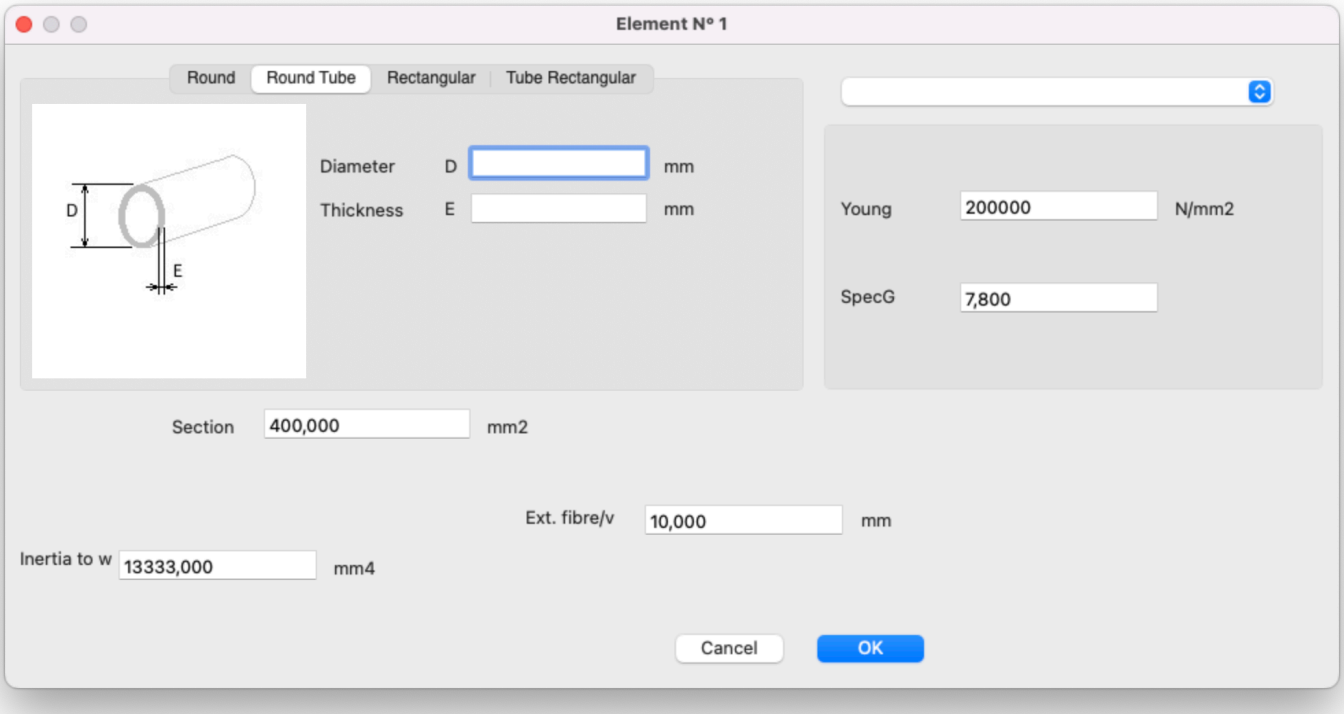
Section : The sectional area of the beam.

Inertia according to w : or quadratic Moment $I_W = S\ D_S\ .\ v^2$.

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

SpecG : The specific gravity of material used. It's ratio between masse for a volume of material and the mass for a same volume of water.

A double click on a line, opens an editing window for the item.

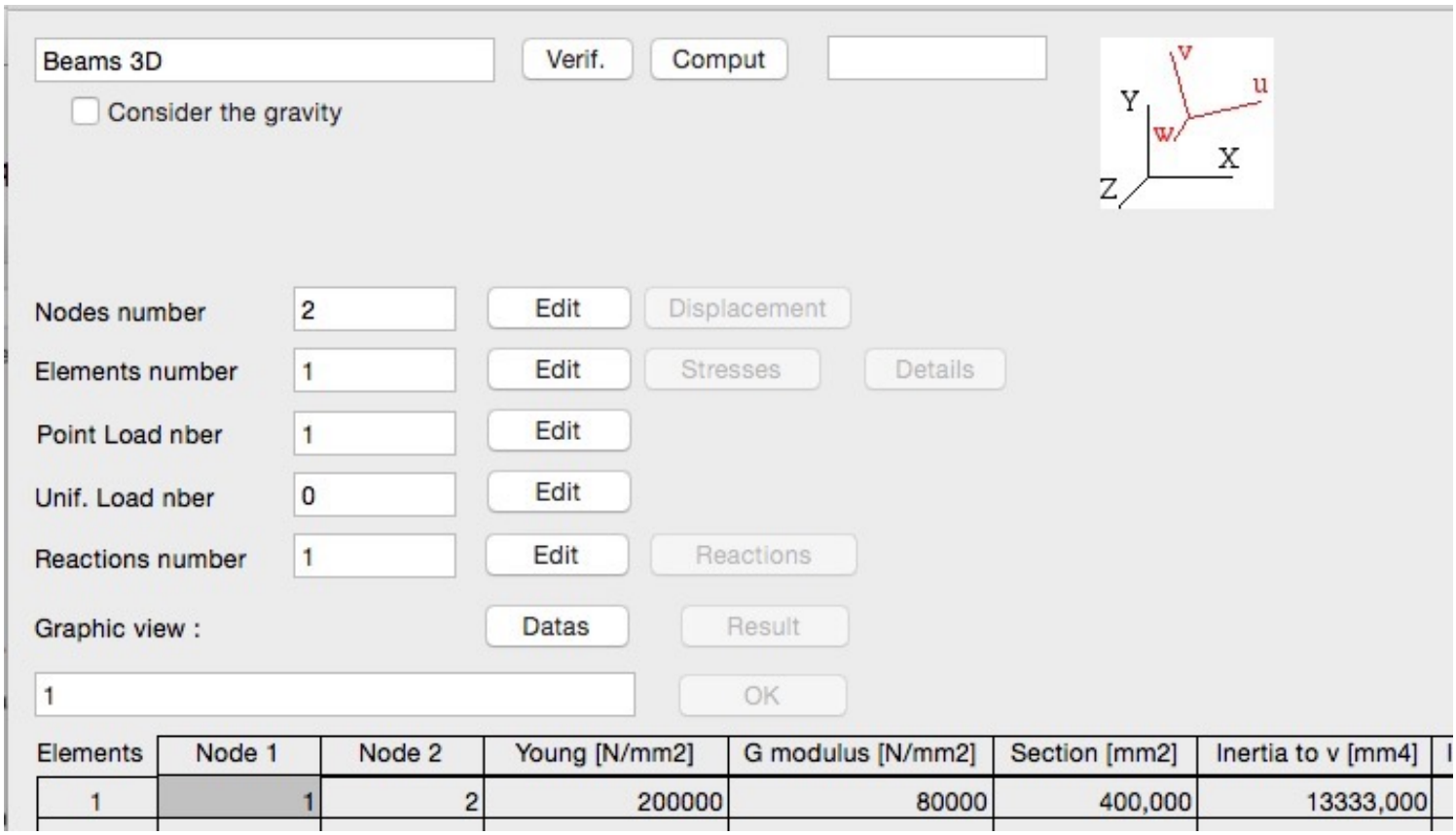


This window lets you choose the shape of the beam and its material.
The other parameters are calculated:

- section
- Inertia following w
- Distance from the outer fiber from the neutral axis.

You can also enter the desired values if necessary.

3.4 / Characteristic of Beams 3D :



Elements	Node 1	Node 2	Young [N/mm2]	G modulus [N/mm2]	Section [mm2]	Inertia to v [mm4]
1	1	2	200000	80000	400,000	13333,000

Elements	Inertia to v [mm4]	Inertia to w [mm4]	Ext. fibre/u [mm]	Ext. fibre/v [mm]	Ext. fibre/w [mm]	SpecG
1	13333,000	13333,000	14,000	10,000	10,000	7,800

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

G modulus (trav.): traverse modulus of elasticity or Coulomb's modulus.

Section: The sectional area of the beam.

Inertia according to v: or quadratic Moment $I_v = S\ D_s.\ w^2$.

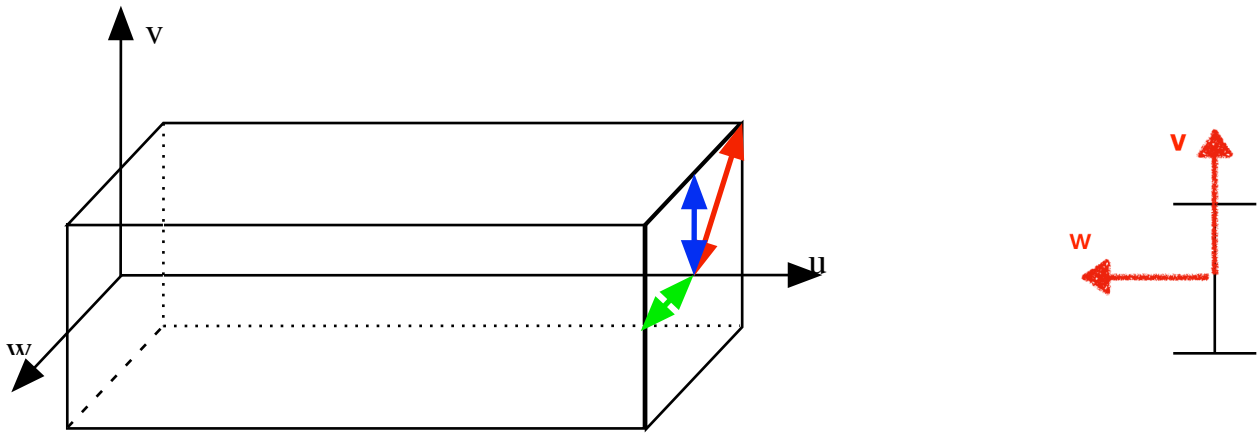
Inertia following w: or quadratic Moment $I_w = S\ D_s.\ v^2$.

Distances between external fibres and neutral fibre: these distances are used for the calculation of stresses.

Ext. Fibre/u : for twist.

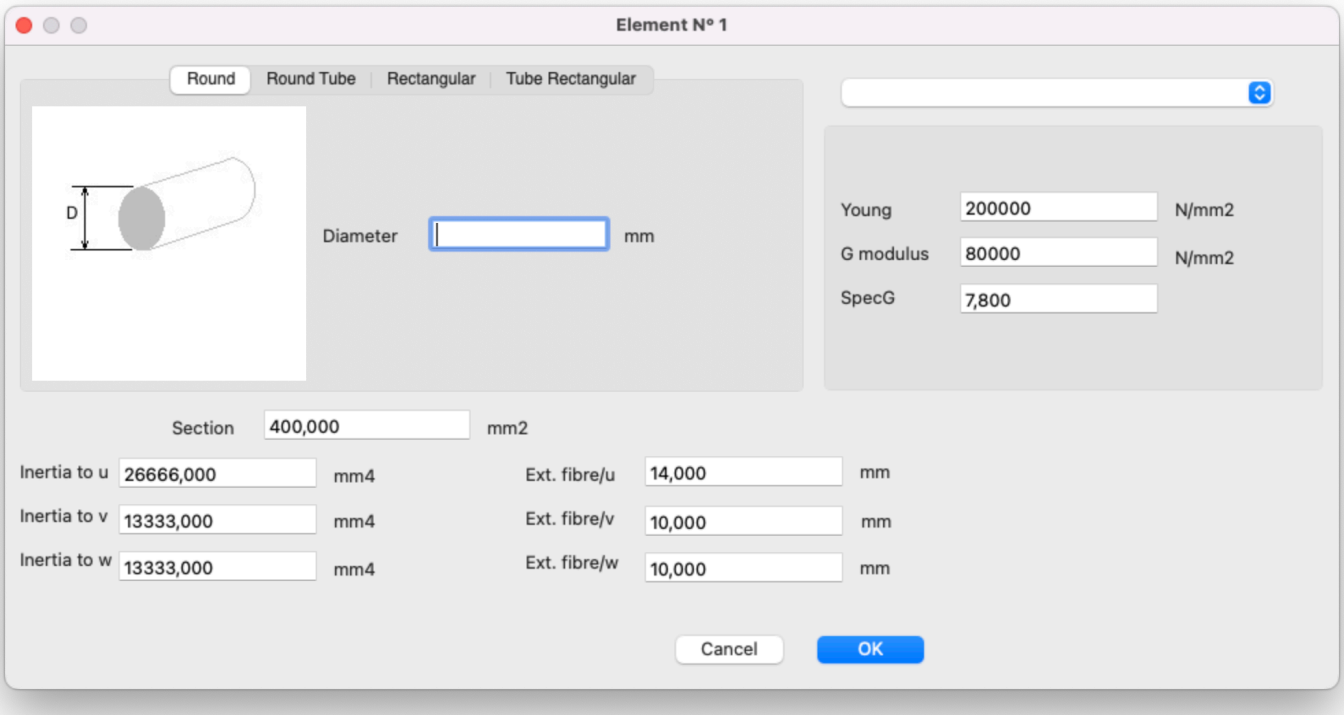
Ext. Fibre/v : for flexion to w axle. Distance between external fibre and neutral fibre following the v axis.

Ext. Fibre/w : for flexion to v axle. Distance between external fibre and neutral fibre following the w axis.



SpecG : The specific gravity of material used. It's ratio between masse for a volume of material and the mass for a same volume of water.

A double click on a line, opens an editing window for the item.



This window lets you choose the shape of the beam and its material.
The other parameters are calculated:

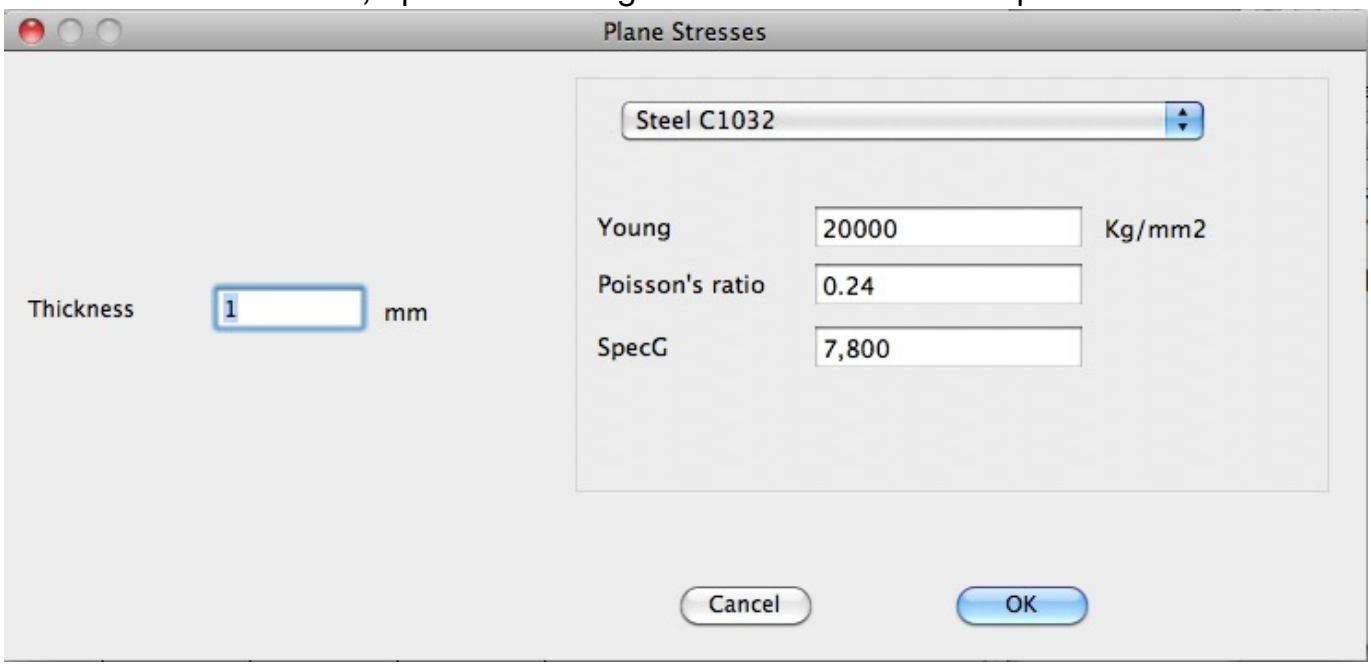
- section
- Inertia following u, v, w
- Distance from the outer fibers from the neutral axis.

You can also enter the desired values if necessary.

3.5 / Characteristics of the Plates :

This item of data entry menu is available only for plate elements.

Double click on a line, opens an editing window features of the plate.



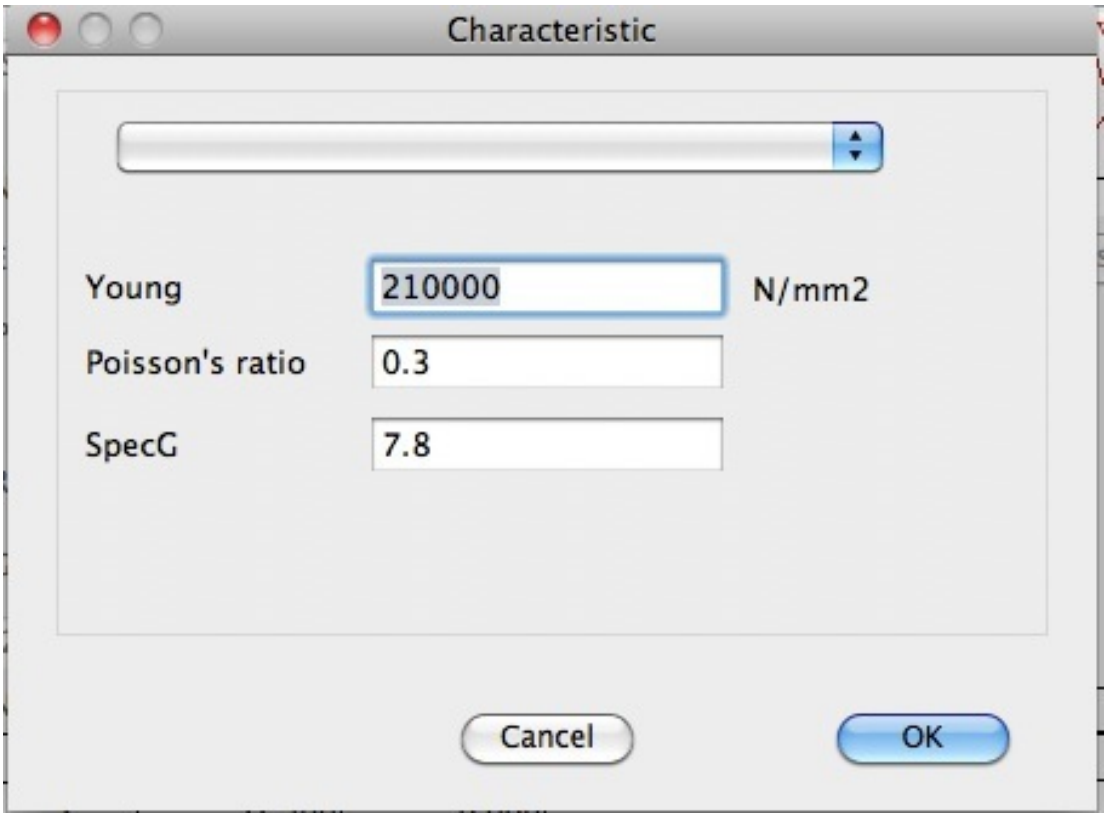
Young's modulus: longitudinal modulus of elasticity of material employed, it is often called ' E'.

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

Thickness: thickness of the plate or the slice considered.

SpecG : The specific gravity of material used. It's ratio between masse for a volume of material and the mass for a same volume of water.

3.6 / Characteristics of revolution Solide :



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

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

SpecG : The specific gravity of material used. It's ratio between masse for a volume of material and the mass for a same volume of water.

3.7 / The 2D Reactions :

Inserted

Reactions	Node	Rx	Ry	RMz
1	1	1	1	1

Simple support

Reactions	Node	Rx	Ry	RMz
1	1	1	1	0

Unidirectional support

Reactions	Node	Rx	Ry	RMz
1	1	0	1	0

Rotation blockage in z axis

Reactions	Node	Rx	Ry	RMz
1	1	0	0	1

3.8 / The 3D Reactions :

Inserted

Reactions	Node	Rx	Ry	Rz	RMx	RMy	RMz
1	1	1	1	1	1	1	1

Simple support

Reactions	Node	Rx	Ry	Rz	RMx	RMy	RMz
1	1	1	1	1	0	0	0

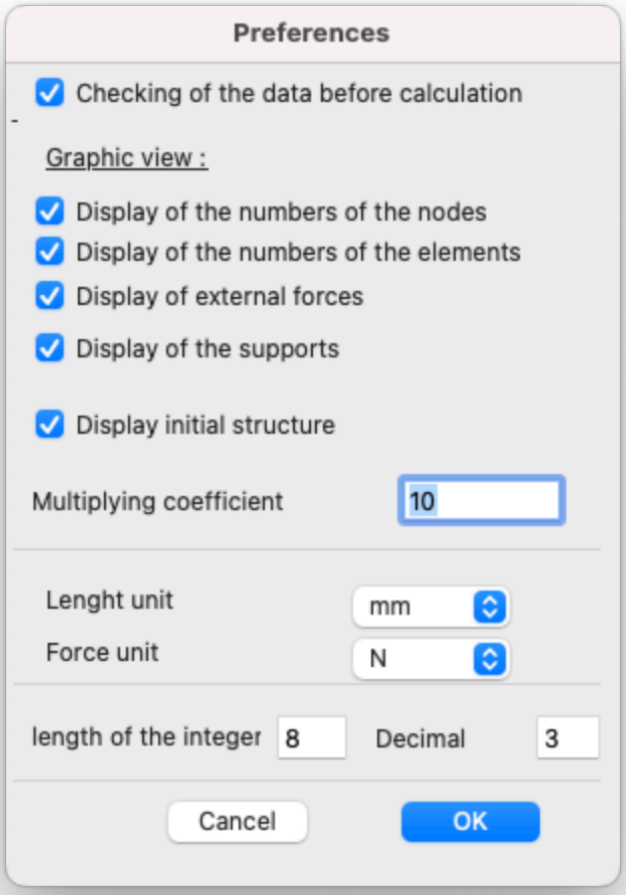
Unidirectional support

Reactions	Node	Rx	Ry	Rz	RMx	RMy	RMz
1	1	0	1	0	0	0	0

Rotation blockage in z axis

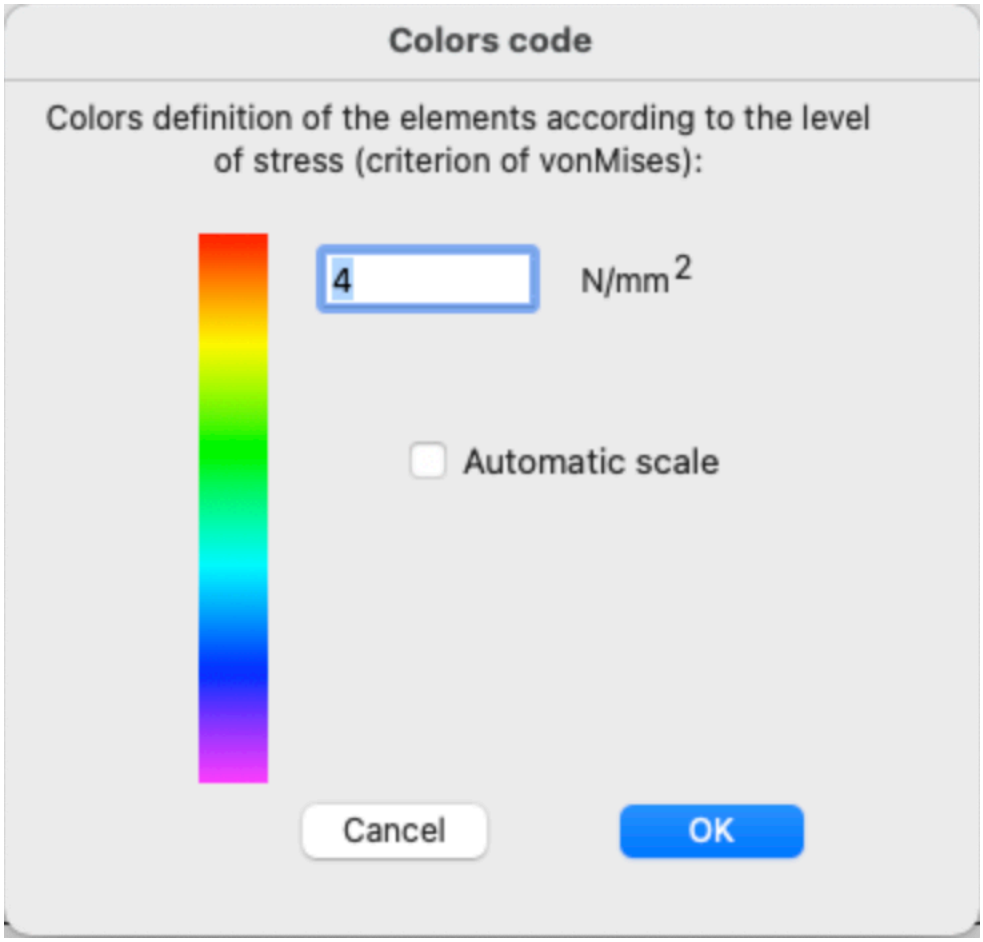
Reactions	Node	Rx	Ry	Rz	RMx	RMy	RMz
1	1	0	0	0	0	1	0

3.9 / 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 shown near the nodes of the structure.
Display of the numbers of the elements: the numbers of the elements are shown near the bars or in the triangles representing 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 gray 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 .
Lenght unit: Select the unit of lenght you use.
Force unit: Select the unit of force you use.
Integer/decimal part length: Defines the format of numbers in tables.

3.10 / The codes colors :

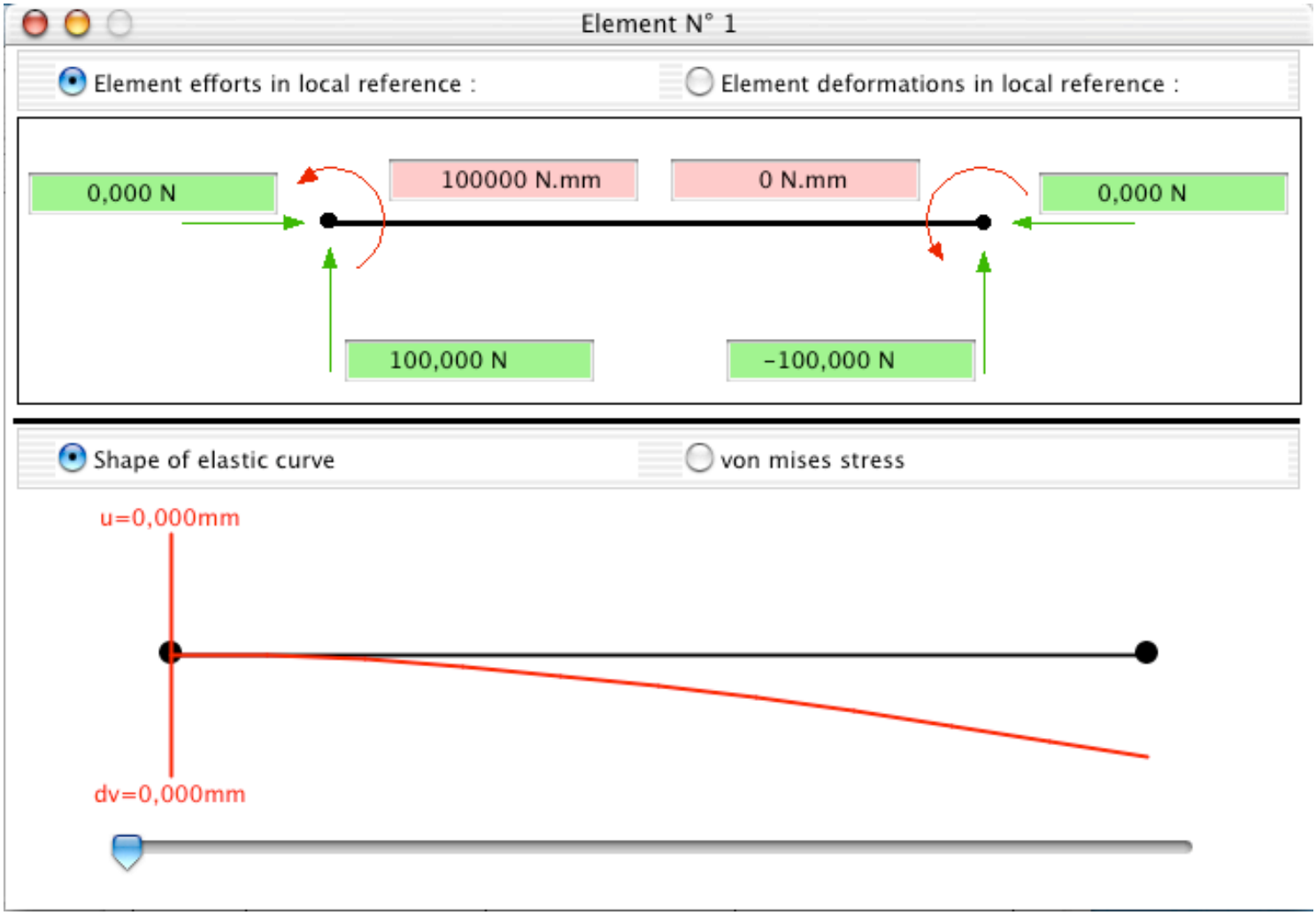


The elements are drawn in the color corresponding to the level of the internal stresses. The program has calculated the absolute value of vonMises stress and then selected the adequate color. In this example:
- the purple elements have an internal stress ranging near to 0.

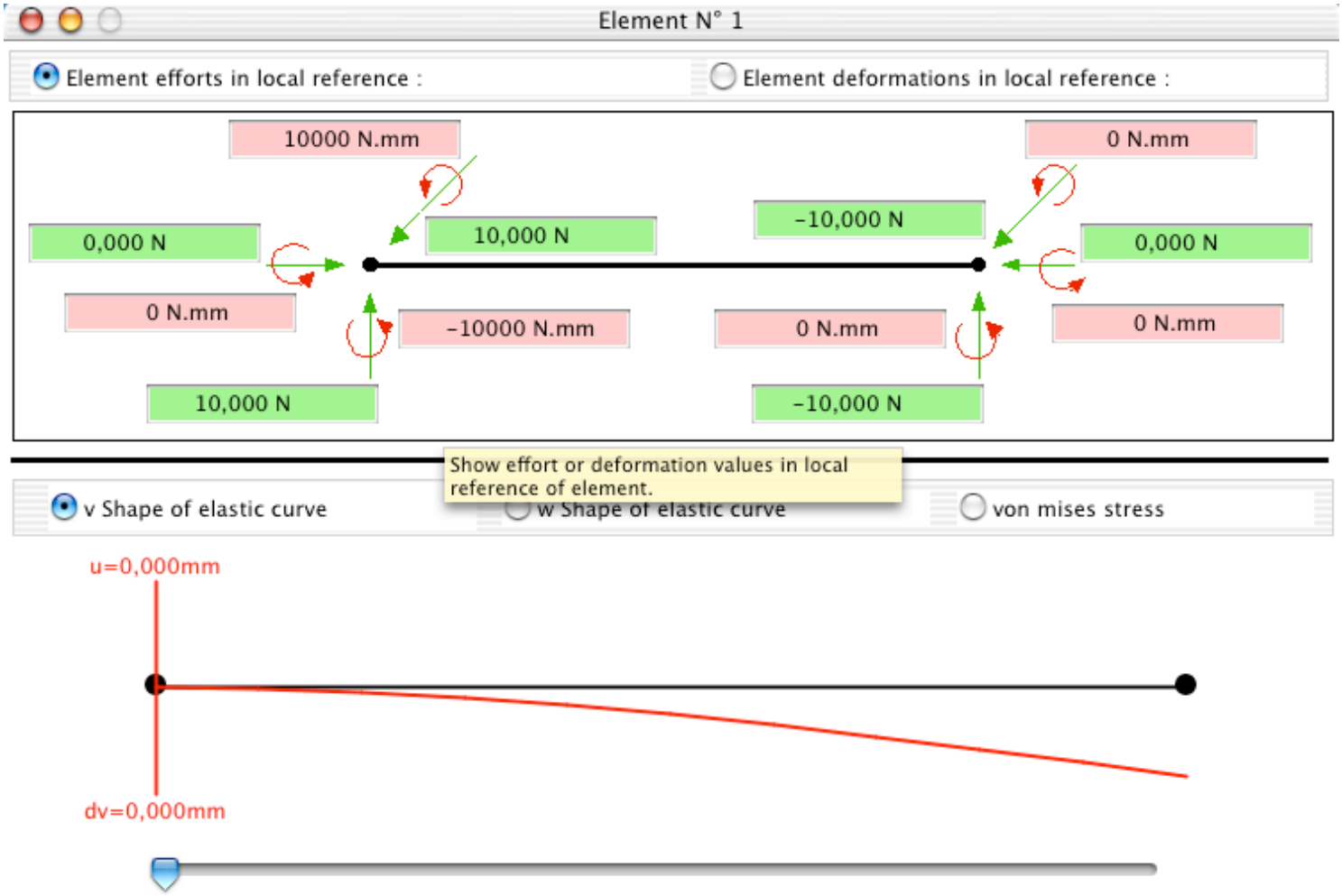
- the red elements have an internal stress maximum.

If “Automatic Scale” is checked, Structurix will evaluate the stresses in all the elements, will retain the maximum stress, to define the range of colors.

3.11 / Screen of details of 2D element :

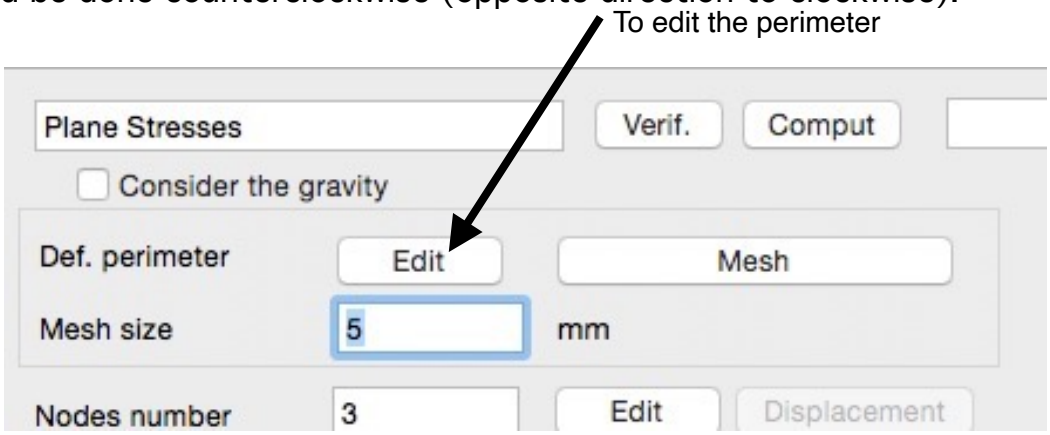


3.12 / Screen of details of 3D element :



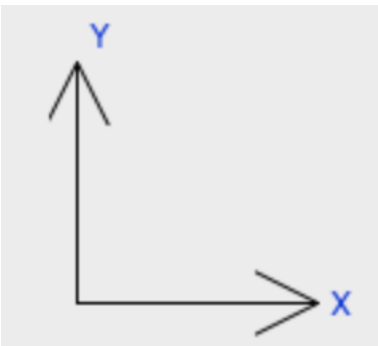
3.13 / Automatic triangular mesch :

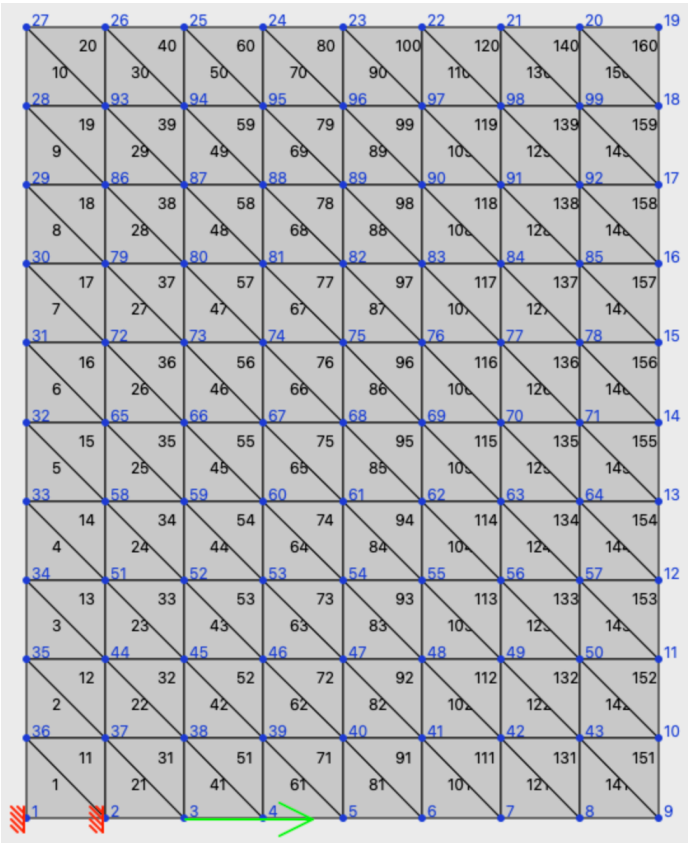
It is possible to generate a triangular mesh automatically. This applies to the problems of plane stress, plane strain and solid of revolution.
The perimeter of the mesh must be described by a succession of points which are given their x, y coordinates. The perimeter thus defined describes the outline of the solid, and should be done counterclockwise (opposite direction to clockwise).



To enter the points that describe the perimeter, you should click on the "Edit" button in the Definition of perimeter. Points are to be listed in the edit table.
Put only the corresponding points to a change of direction, it is not necessary to indicate the points for each step of the mesh, Structurix add the points necessary to meet the size of the mesh.
The last line should repeat the coordinates of the first point, that Stucturix know that the description is complete.
The example below shows how to describe a rectangle, and the result obtained.

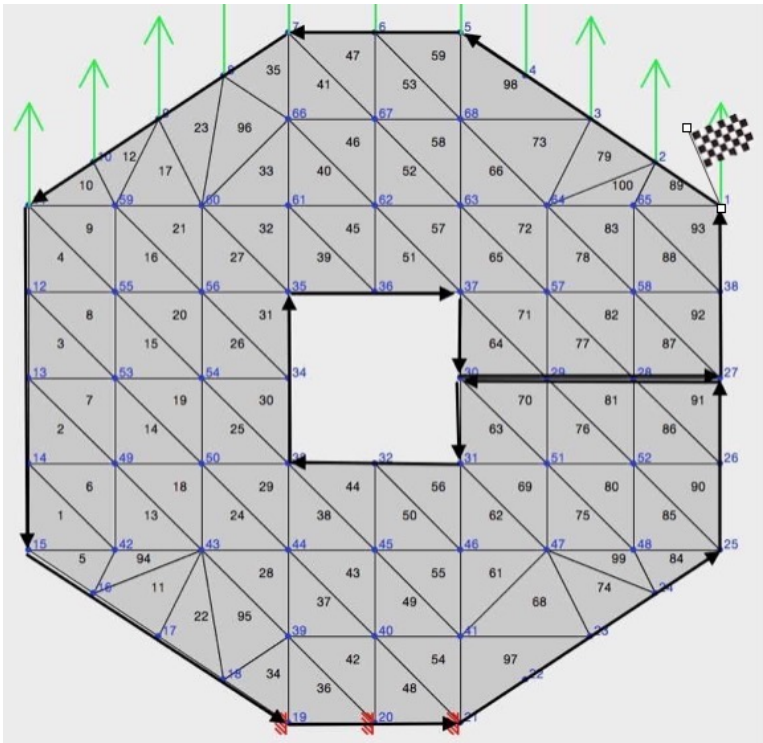
Périmètre	x [mm]	y [mm]
1	10,000	0,000
2	50,000	0,000
3	50,000	50,000
4	10,000	50,000
5	10,000	0,000





In the case of a hole, must reach the hole area and the join perimeter with the same sequence of points. See example of hole in octagon:

Perimeter	x [mm]	y [mm]
1	20,000	10,000
2	5,000	20,000
3	-5,000	20,000
4	-20,000	10,000
5	-20,000	-10,000
6	-5,000	-20,000
7	5,000	-20,000
8	20,000	-10,000
9	20,000	0,000
10	5,000	0,000
11	5,000	-5,000
12	-5,000	-5,000
13	-5,000	5,000
14	5,000	5,000
15	5,000	0,000
16	20,000	0,000
17	20,000	10,000



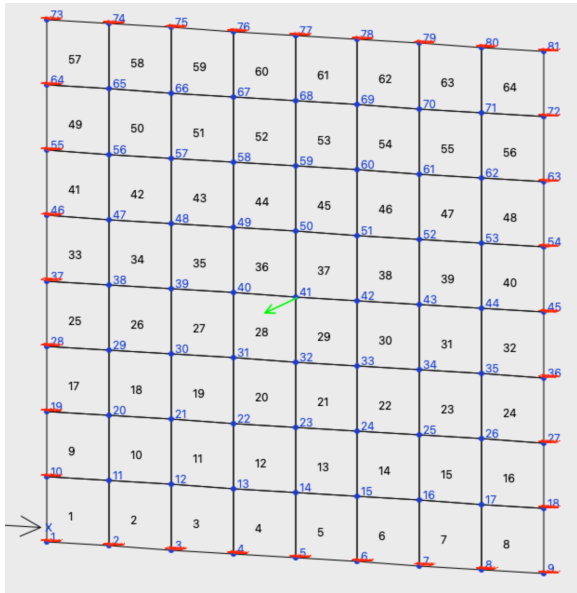
3.14 / Automatic rectangular mesh:

It is possible to generate a rectangular mesh automatically. This is valid for flat bending problems.

The perimeter of the mesh must be described by a succession of points, of which one gives their coordinates x, y. The perimeter thus described defines the contour of the solid, and must be done counterclockwise (anti-clockwise).

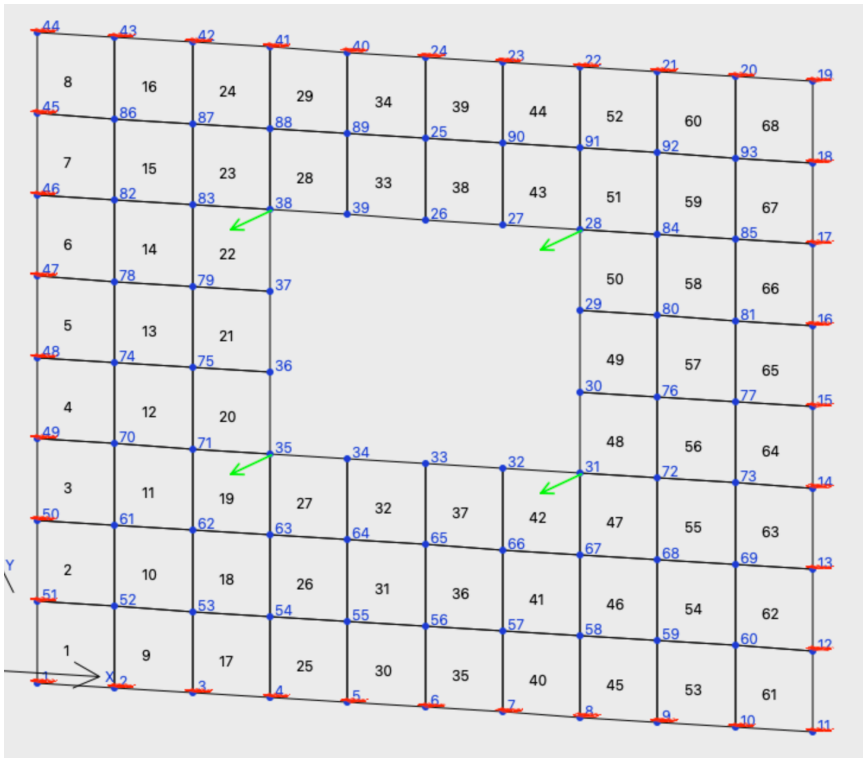
To enter the points which describe the perimeter, click on the "Edit" button of the Perimeter definition. The points are to be listed in the editing table.
Only put the points corresponding to a change of direction, it is not necessary to indicate the points for each step of the mesh, Structurix will add the points necessary to respect the size of the mesh.
The last line should repeat the coordinates of the first point, so that Structurix knows that the description is complete.
The segments must be horizontal or vertical, they cannot be oblique.
The length of the segments must be a multiple of the mesh pitch.
The example below shows how to describe a rectangle, and the result.

Perimeter	x [mm]	y [mm]
1	0,000	0,000
2	80,000	0,000
3	80,000	80,000
4	0,000	80,000
5	0,000	0,000



In the case of a hollow area, you have to reach the hollow area and reach the perimeter with the same succession of points. See example of the hollow octagon:

Perimeter	x [mm]	y [mm]
1	0,000	0,000
2	100,000	0,000
3	100,000	80,000
4	50,000	80,000
5	50,000	60,000
6	70,000	60,000
7	70,000	30,000
8	30,000	30,000
9	30,000	60,000
10	50,000	60,000
11	50,000	80,000
12	0,000	80,000
13	0,000	0,000



When the description is finished, you must launch the generation of the mesh by clicking on the "Mesh" button.
The different steps in Structurix are:
- Calculation of the perimeter by adding the nodes on the perimeter, to respect the size of the mesh.
- Addition of points on the whole rectangle which covers the perimeter, respecting the size of the mesh.
- Removed points that are outside the perimeter.
- Creation of elements using the available points.

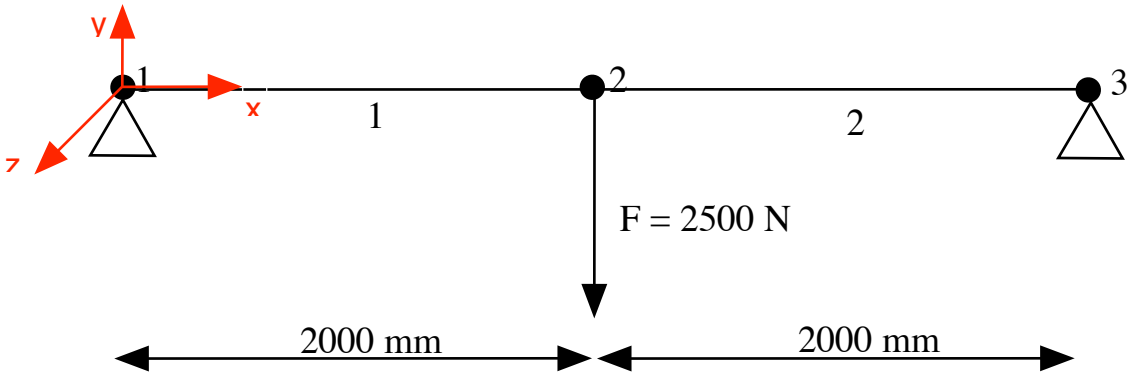
At the end of the generation of the mesh. The points are copied into the nodes array, the elements are copied into the elements array.
You have to define the supports and the forces applied to the nodes.

4 / Tutorial with a simply sample :

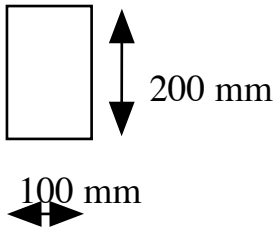
4.1 / Description of problem :

A wood beam with section 200 mm x 100 mm is placed on two supports distant by 4 meters. In the middle, we put one load of 2500 Newton.

Schematic representation :



Caracteristics of two wood beams



E = 10000 N/mm2

4.2 / Setting the nodes :

Nodes	x [mm]	y [mm]
1	0,000	0,000
2	2000,000	0,000
3	4000,000	0,000

The node n° 1 have for coordinate (0,0).
The node n° 2 have for coordinate (2000,0).
The node n° 3 have for coordinate (4000,0).

4.3 / Setting the elements :

Elements	Node 1	Node 2	Young [N/mm2]	Section [mm2]
1	1	2	10000	20000,000
2	2	3	10000	20000,000

Elements	2]	Section [mm2]	Inertia to w [mm4]	Ext. fibre/v [mm]	SpecG
1	00	20000,000	66666666	100,000	7,800
2	00	20000,000	66666666	100,000	7,800

The first element is to node 1 to node 2.
It has a Young's modulus = E = 1000 N/mm2, because it is wood oak.
Its section is 200 x 100 = 20000 mm2.
The inertia of w is calculated by the formula $bh^3 / 12 =$
$100 * 250^3 / 12 = 66666666\ mm^4$.
The position of external fiber to axle v is 200/2 = 100 mm.

4.4 / Setting the external force :

Ext. Force	Node	Fx [N]	Fy [N]	Mz [N.mm]	
1	2	0,000	-2500,000	0	

One force is applied on node 2. Its X componant is null, its Y componant is egal to -2500 N. It is negative because it is oriented to down.
There is no couple therefore Mz = zero.

4.5 / Setting the support reactions :

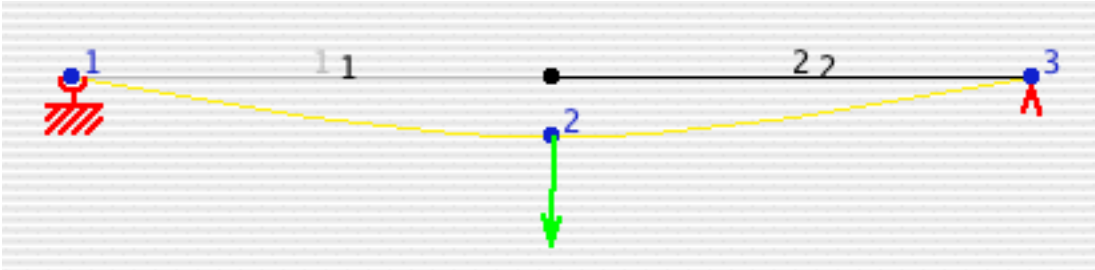
Reactions	Node	Rx	Ry	RMz
1	1	1	1	0
2	3	0	1	0

There are two reactions, one is applied on node 1 and the other acts on node 3.

To define a support, you must specify by 0 or 1 the type of behavior in the direction.
In this case, we have only one reaction on the Y direction. You must put 1 in Ry for each support.
Our systeme is now placing on two supports, but no blocking on X direction. If you comput, a error message will said “Shortage of blockage of structure following X”
As a logical deduction we know that X efforts are null, but the program doesn’t know in advance that X component will be null. For this reason, you must block the structure in the X direction on one support.

4.6 / Computing and results :

Displts	dx [mm]	dy [mm]	dMz
1	0,000	0,000	-0,004
2	0,000	-5,000	0,000
3	0,000	0,000	0,004



5 / Characteristics of some materials:

E [N/mm^2 ou MPa]: Young N or modulus of elasticity.

G [N/mm^2 ou MPa]: traverse modulus of elasticity.

υ : Poisson's number.

S1 [N/mm^2 ou MPa]: elastic limit in traction.

specG = Specific Gravity.

Material	E	G	υ	s1	SpecG
Iron	200000	80000	0,24	200	7,80
Steel XC10	216000	86400	0,29		7,80
Steel C32	200000	80000	0,24	370	7,80
Steel C45	200000	80000	0,24	400	7,80
Steel 35NCD4	200000	80000	0,24	900	7,80
Steel 45SCD6	220000	88000	0,29	1450	7,80
Stainless steel 18.10	203000	81200	0,29	200	7,90
Current gray cast iron	90000	36000	0,29	190	7,20
Titane TA6V	105500	42200	0,34	300	4,50
Alloy titanium Ti 4 Al 4	115000	46000	0,34	900	4,42
Aluminium	70500	28200	0,34	150	2,70
Alloy AU 4 G	72000	28800	0,32	200	2,80
Alloy AU 2 GN	75000	30000	0,34	370	2,80
Zicral AZ 8 GU	72000	28800	0,34	550	2,80
Copper	100000	40000	0,33	180	8,90
Brass Cu Zn 5	125000	50000	0,38	200	8,30
Brass Cu Zn 40	105000	42000	0,34	220	8,30
Bronzes ordinary	106000	42400	0,31	240	8,40
Bronze with beryllium	130000	52000	0,34	800	8,25
Beryllium	300000	120000	0,05	300	1,85
Magnésium	46000	18400	0,34	180	1,74
Zinc	130000	52000	0,21	120	7,15
Nickel	205000	82000	0,31	300	8,90
Concrete	27000		0,20	1,2tens./8comp.	2,20
Granite	60000		0,27	65-150	2,70
Wood oak	10000	500		12	0,80
Wood resinous	17000	1000	0,45	11	0,60
Plexiglass	2900	1160	0,40	80	1,80
Glass	60000	24000	0,20	60	2,50
Araldite	3000	1200	0,40	70	1,15
Rubber	2	1	0,50		0,98

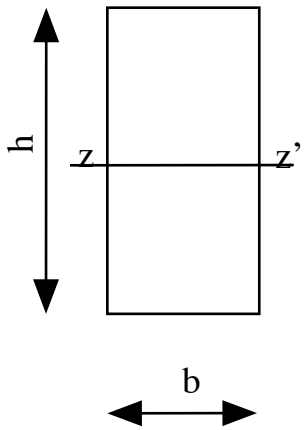
The file material.csv contains the characteristics of different materials are shown in the windows of the characteristics of the elements. this file must be in the folder containing the application. if absent, a shortlist is been presented.

in U.S. units

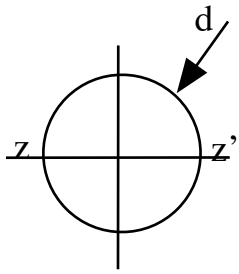
E = Young Modulus of Elasticity (p.s.i.)
G = Traverse Modulus of elasticity (p.s.i.)
υ = Poisson's number.
s1 = elastic limit in traction (p.s.i.)
specG = Specific Gravity
r : density (p.c.i.)

Material	E	G	υ	s1	SpecG	r
Iron	30.0×10^6	11.6×10^6	0.24	29000	7.8	0.28
Steel C10xx	31.3×10^6	12.5×10^6	0.29		7.8	0.28
Steel C1032	30.0×10^6	11.6×10^6	0.24	53600	7.8	0.28
Steel C1045	30.0×10^6	11.6×10^6	0.29	58000	7.8	0.28
Steel 35NCD4	30.0×10^6	11.6×10^6	0.24	130500	7.8	0.28
Steel 45SCD6	31.9×10^6	12.7×10^6	0.28	210300	7.8	0.28
Stainless Steel 18.10	29.4×10^6	1.18×10^6	0.29	29000	7.9	0.29
Cast Iron (current gray)	13.1×10^6	5.22×10^6	0.29	27550	7.2	0.26
Titane TA6V	15.3×10^6	6.12×10^6	0.34	43500	4.5	0.16
Titanium alloy Ti 4 Al 4	16.7×10^6	6.67×10^6	0.34	130530	4.42	0.16
Aluminium	10.2×10^6	4.09×10^6	0.34	21750	2.7	0.10
Alloy AU 4 G	10.4×10^6	4.18×10^6	0.32	29000	2.8	0.10
Alloy AU 2 GN	10.9×10^6	4.35×10^6	0.34	53660	2.8	0.10
Zicral AZ 8 GU	10.4×10^6	4.18×10^6	0.34	79770	2.8	0.10
Copper	14.5×10^6	5.80×10^6	0.33	26100	8.9	0.32
Brass CU Zn 5	18.1×10^6	7.25×10^6	0.38	29000	8.3	0.30
Brass Cu Zn 40	15.2×10^6	6.09×10^6	0.34	31900	8.4	0.30
Bronzes (ordinary)	15.4×10^6	6.15×10^6	0.31	34800	8.25	0.30
Bronzes (with Beryllium)	18.9×10^6	7.54×10^6	0.34	116030	1.85	0.30
Beryllium	43.5×10^6	17.4×10^6	0.05	43500	1.74	0.07
Magnesium	6.67×10^6	2.67×10^6	0.34	26100	1.74	0.06
Zinc	18.9×10^6	7.54×10^6	0.21	17400	7.15	0.26
Nickel	29.7×10^6	11.9×10^6	0.31	43500	8.3	0.32
Concrete	3.92×10^6	1.3×10^6	0.20	174(tensile) 1160(compression)	2.2	0.08
Granite	8.70×10^6		0.27	9425-21750	2.7	0.10
Wood Oak	1.45×10^6	72500		1740	0.8	0.03
Wood resinous	2.47×10^6	145000	0.45	1600	0.6	0.02
Plexiglass	0.42×10^6	168000	0.40	11600	1.8	0.07
Glass	8.70×10^6	3.48×10^6	0.20	8700	2.5	0.09
Araldite	0.44×10^6	174000	0.40	10150	1.15	0.04
Rubber	290.1	116	0.50		0.98	0.04

6 / Inertia :

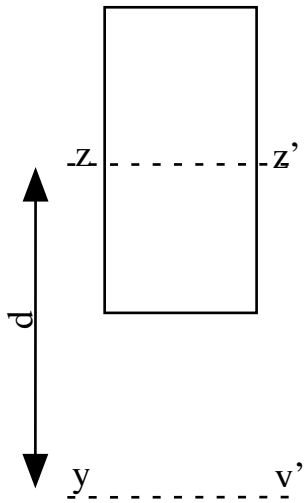


$$Izz' = \frac{bh^3}{12}$$



$$Izz' = \frac{\pi d^4}{64}$$

Huygens Theorem



$$Iyy' = Izz' + S d^2$$

with S: surface

7 / the author :

Program written by Yannick CALLAUD with Xojo.

yannick.callaud@laposte.net
http://y.callaud.free.fr

Thank you to Fabrice JAME - AMBARES (33) France, who induced me to create a WinTel version. He also suggested many improvements of benefit for all users.

8 / Structurix evolutions :

- version 3.7.1 of 18 December 24: Compilation with new version of Xojo which allows MultiCore. Which allows calculations 20 times faster. Update of some routines to guarantee compatibility with future versions of Xojo.
- version 3.7.0 of 17 July 24: Saves the results to avoid a sometimes lengthy re-calculation to display the results.
  Updated many routines to ensure compatibility with future versions of Xojo.
- version 3.6.0 of 9 February 24: Displays context-sensitive help for the Max Constraint edit field for setting the color scale.
  Added possibility of copying one of the result tables to the clipboard: deformation, stresses, reactions. For example: in order to transfer the results into a spreadsheet.
  Fixes a rounding issue in the coordinates of points generated when automatically meshing rectangular elements.
  Fixes a defect in the graphical display of reactions for rectangular plate problems. This problem appears for cases of small dimensions.
  Fixed a problem that showed deformations in x and y on rectangular plate problems, while only the deformations in z are calculated.
  Added indeterminate progress bar in the information window, when creating elements and during calculation.
  Improved the display of rectangular and triangular element numbers on the data graph.
- version 3.5.3 of 20 July 23: Correction to display the list of materials in the correct language: French for France, English for UK and others.
- version 3.5.2 of 31 May 23: Bug fixes, reported by Lionnel Angélidès:
  - the bar section can be edited in the characteristics window.
  - the characteristics of the beams can be edited in the characteristics window.
- version 3.5.1 of 7 November 22:
  The default language is now English instead of French.
- version 3.5.0 of 18 October 22:
  Bug fixes:
  - Display of the 3D marker on the main window.
  - Removed beep when entering the characteristics of an element when a field is empty.
  - Shape of the deformation of the 2D beams.
  sAdded element "Rectangular tube" for the calculation of the characteristics of an element.
- version 3.4.1 of 14 October 22: Fixed display bug:
  - logos in buttons for graphics animation.
  - images in some windows.
- version 3.4.0 of 1 April 22: Added hourglass cursor when meshing.
  - Improved the option to calibrate the color scale according to the maximum stress.
  - Re-positioning window so that it is visible if it is displayed outside the screen.
  - Acceleration of the display in graphic form.
  - Added display of stress level in the element.
  - Added messages specifying the progress of the mesh.
  - Improvement of the mesh, in particular on the shape of the triangular elements. And

added triangle shape optimization routine.
- Added display of time spent meshing.
- Document files can be opened with a double-click.
- Fixed display bug on 2D Beams in deformed position if gravity has been taken into account.

- version 3.3.2 du 6 March 22: Fixed bug in the automatic triangular mesh, which no longer worked. Added "Beep" at the end of the automatic mesh.
  The maximum stress was not displayed correctly in the graphical visualization of the results.
- version 3.3.1 of 6 June 21: Better update of the max constraint in result window.
  More explicit error message when the structure is not blocked enough.
- version 3.3.0 of 02 March 21:
  Bending of the plates: Removal of the column containing the z coordinates of the nodes. And, addition function of mesh by rectangle for the bending of the plates.
  Added option to automatically set the color scale.
- version 3.2.2 of 16 February 21: Correction of a calculation error for plates subjected to bending, when gravity was taken into account.
- version 3.2.1 of 13 February 21: The calculation of the plates, did not take into account gravity. Bug reported by Vaidas Baranauskas.
  Structurix gives control to the system during the calculation, it no longer seems blocked.
  The calculation time is set to zero when a new calculation is started.
  Displacements by nodes are displayed at the end of the calculation.
  Added display of node numbers to the plate result graph.
- version 3.2.0 of 4 December 20: Evolution of the window for the definition of the color of the elements according to its stress.
- version 3.1.4 of 11 Aug 19: Added refresh of the array of elements after the modification of the characteristics.
- version 3.1.3 of 31 May 17 : Amendment of the help for the taking into account of gravity. The Y axis is the vertical axis except for the plates whose vertical axis follows the Z axis. Error reported by Lionel Byledbal.
- version 3.1.2 of 24 Feb 17 : Fixed bug, problem of display of the forces distributed on the elements beam 3D. The display of the sections of the 3D beams will no longer be deleted if the number of the elements is no longer required.
- version 3.1.1 of 12 Jan 16 : Fixed bug that occurred when saving Bars 2D, Bars 3D, beams 2D, beams 3D, flexion plate.
- version 3.1.0 of 13 Jul 15 : Adding automatic meshing for Stresses Plane, plane deformation, and solid of revolution. The triangular elements are now dimmed in the graphical representation of data to visualize the holes or gaps.
  If adding a row, the array automatically scroll and the cell of the first column is selected for immediate entry.
  Copy and paste now works with Number. Fixed error that prevented the calculation of stresses in plane strains.
- version 3.0.7 of 20 Dec 14 : Best proportion for graphical display 2D.
  Recalculation of the display if the graphics window is re-sized.
- version 3.0.6 of 27/11/14 : Center the graphic display of data and results.
  Bug fixes for the recording of Young's modulus for items flat Constraints and Plane strain.

- version 3.0.5 of 10/11/14 : Adds the extension ".dat" when saving the file. Following remark Richard Lance.
- version 3.0.4 of 22/09/14 : Fixed error for axial stresses
  2D and 3D beams. Error in the use cases of distributed forces or taking account of gravity. Bug discovered and reported by Xavier Dumont.
- version 3.0.3 of 9/08/14 : Adding time display of the computing time.
- version 3.0.2 of 10/03/13 : Fixed calculation error on the constraint vonMises of elements rectangular plates. Errors found and reported by Gerard Lachenal.
- version 3.0.1 of 6/01/13 : Fixed bug, the button of validation of data entry was not active. Bug discovered and announced by Michel GAUBERT.
- version 3.0.0 of 21/10/12 : Fixed bug on copy/paste on list.
  Material are on correct language.
  Add element «Solid of revolution».
- version 2.9.5 of 23/09/12 : Fixed bug on the 2D graphic for drawing effort.
  Thank you to Xavier Dumont letting me discover.
- version 2.9.4 of 23/10/11 : Fixed bug to allow copy and paste values from a spreadsheet. This facilitates the construction of the structure.
  The window no longer displays inappropriate on the details of the element in the case of flexion plates.
- version 2.9.3 of 03/09/11 : Fixed bug on graphical representation of forces.
  The copy / paste can be used to edit nodes and elements.
  Le copier/coller est utilisable pour éditer les noeuds et les éléments.
- version 2.9.2 of 17/07/11: Bugs correction.
- version 2.9.1 of 14/07/11: In the editing window of the characteristics of elements, the calculation of inertia and sections is done automatically.
- version 2.9.0 of 19/06/11: The column width is based on the format defined in the preferences.
  Improving the representation of forces distributed in the graphs.
  Adding an editing window elements to facilitate the calculation of inertia, section, and distance from the external fibers from the neutral axis.
  The file contains the material.cvs characteristics of the materials offered in the editing windows of the characteristics of the elements.
- version 2.8.1 on 13 june 11 : The cell editing numeric values to be written digital format defined in the preferences.
  Improving the representation of forces and reactions in graphical representations.
  Adding a reference X, Y, Z, in graphical representations.
- version 2.8.0 : Addition of the possibility of copying and of sticking parts of table of figures. That makes it possible to stick data coming from a spreadsheet.
- version 2.7.1 : Fixed bug representation of the external forces at 3D visualization.
  Thank you to Martin Ruiz for pointing.
  Fixed bug representation of the section `I` in 3D visualization. The rotations in 3D view are controlled by a click-drag and not by optiion-scroll.
- version 2.7.0 : Addition in the preferences, the possibility of fixing the number of decimal.
  Bug correction of calculation in rotations of the elementary rigidity matrices for the constitution of the matrix of total rigidity. Thank you in Martin RUIZ to have identified it.

In graphic window. Buttons allow, the zoom, rotation and it displacement of the graph, posting or not, of the numbers of the nodes, the elements, the forces and the reactions.
Better representation of the forces in 3D.
The horizontal scroll was added in the main window.

- version 2.6.4 : Corrected bug which prevented the display of graphic result of bars 3d. Thanks to François Rimbert.
- version 2.6.3 : When you print, It is possible to delimit the pages you want.
- version 2.6.2 : Corrected bug when you print the result and when the number of ponctual effort is null. Thanks to Christophe.
- version 2.6.1 : Corrected bug when you print and when the number of ponctual effort is null.
- version 2.6.0 : Added flexion plate element.
  In plane deformations and plane stresses, Structurix take the maximum stresses instead of the stresses of the first node.
  The wheel mouse work.
- version 2.5.5 : Corrected bug on decimal separator.
- version 2.5.4 : Corrected bug on stresses calculation.
- version 2.5.3 : Supression of error message when you comput without ponctual effort.
  Modification calculation of uniform load to avoid incoherences on results.
  Correct bug on displacements calculation and efforts of Beam 3D elements with uniform load.
- version 2.5.2 : Corrected bug when you would delete a reaction.
- version 2.5.1 : Corrected the density by SpecG in English version.
  Thanks to Gabriel Fuentes.
- version 2.5.0 : Deleted column InertieX, no necessary because it is calculed with InertieY and InertieZ.
  Added 3D reference in main window.
  Added notion local (u,v,w) and global (x,y,z) reference.
- version 2.4.1 : Added IPN section drawing of 3D beams to show orientation of beam in global reference.
  Added table of uniform load in printing.
  Correct error of reaction when they are a load on support.
  Correct graphic display on 2D beams with uniform load.
  Correct calculation stress on 3D beam.
- version 2.4.0 : Added curves of deformation and von Mise stress in “détails” window. A few bugs corrected.
- version 2.3.0 : Added “details” window for visualisation of deformations and efforts by element in own local reference.
- version 2.2.0 : Modification of the support drawing for a better representation.
  In PC version, the file extension is now added when you save your document.
- version 2.1.0 : Added balloon help. Added verification of values to choose the color of element when you push OK.
- version 2.0.1 : Added drawing of uniform load in graphic views.
  Multiplying coefficient is selected when you open the preferences window.
  Bugs correction : Error when the beams are drawing in graphic view, impossibility

to add distributed load after a file is open.

- version 2.0.0 : Added possibility to take in account the influence of gravity.
  Added possibility to put distributed load on beam element.
  Correct a bug when the program creates the index matrice for the reactions.
  Improvement of structure verification before computing.
  Added verification of structure before displaying the graphic.
  Added dialog during calculation.
  Added verification of enumerate direction of triangular element.
- version 1.4.0 : Possibility to select futher Cells to allow a down copy. It is to avoid several data entry. Correct a bug in result beams 2D. (bad deformation)
  Added fork tool Icon with mesh.
- version 1.3.0 : Rewrite the print aspect grace at Fabrice JAUME's advices.
  All tables are printed in same time.
- version 1.2.0 : Addition unity to avoid confusions.
  Orientation of supports for a better logic.
  Rewrite the language manager (English, French). To simplify the work of programmer when structurix is updated. It is non apparent by the user.
  Add a specific message when the structure is not blocked for all directions.
- version 1.1.1 : Nodes and their number are displayed blue.
- version 1.1.0 : Automatic opening document by a double clic.
- version 1.0.0 : First version placed at the disposal of the public.
  Structurix has been rewritten in Basic with C++ sources of ElemFin.
  Addition of ElemFin :
  - The graphic window can be resized.
  - The size and position of windows are recorded in pref file.

9/ Errors :

My English is not very good. If you find an error, please send me the correction by email.
Corrected by Gabriel Fuentes, 27 November 2003.