NUMERO: 2308A ANNO: 2018

A P P U N T I

STUDENTE: Chiforeanu Loredana

MATERIA: Structural Mechanics II - Lusas guide - Prof. Cornetti -Sapora.pdf

POLITECNICO DI TORINO

MASTER COURSE IN CIVIL ENGINEERING

Course of STRUCTURAL MECHANICS II , YEAR 2017/2018

Lusas notes

How to analyse a structure in Lusas step by step

Professor: **Cornetti Pietro**
Author: **Loredana Mihaela Chiforeanu**

In this report you can find:

- exercises solved in class
- explanations step by step
- images of each part and final results
- examples of exam exercises

My other notes of Master courses:

- *Theory of Structural Mechanics II + Exam 2018*, prof. P.Cornetti / Sapora, year 2017/18
- *Theory and design of reinforced and prestressed concrete structures*, prof. A.P. Fantilli, year 2017/2018
- *Theory of Hydrology*, prof. S. Tamea, year 2017/2018
- *Exercises of Hydrology*, prof. S. Tamea, year 2017/2018
- *Exercises of Construction of Roads, Railways and Airports*, prof. E. Santagata / Dalmazzo/ Riviera, year 2017/18

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 3 di 104

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 5 di 104

- **Intro: calculation of solicitations' diagrams in a given thin beam ("trave sottile")**

The solution of the problem is an approximation because a closed form solution doesn't exist.

Since the 60's of the last century, the finite element method is the most used.

This method consists in discretization and interpolation:

- discretize the structure to simplify it introducing a finite number of nodes and a finite number of elements: from infinite degrees of freedom we pass to a finite number of degrees of freedom: this is done through a mesh;

- the calculation is made on the nodes, in terms of displacements, so we find the nodal displacements. Then from displacements we calculate stresses and other quantities;

- we pass from partial differential equations to ordinary differential equations, having a much easier solution.

Izz is the moment of inertia in the vertical plane around x axis and along z axis.

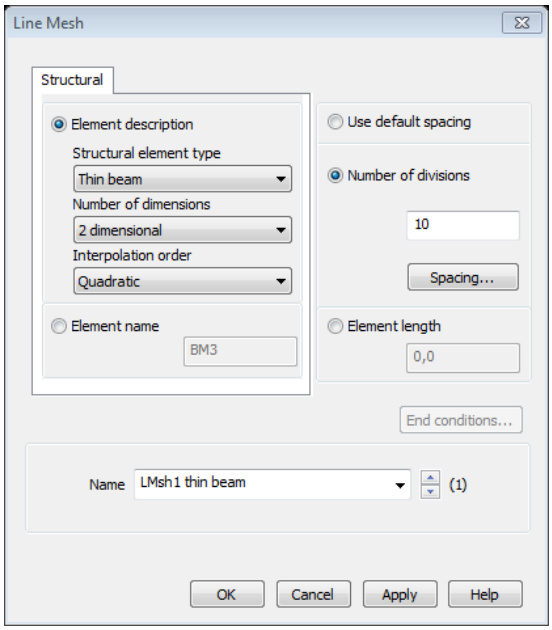
Mz is a linear bending moment and Ty is its first derivative (shear constant).

The distribution of τxy is quadratic.

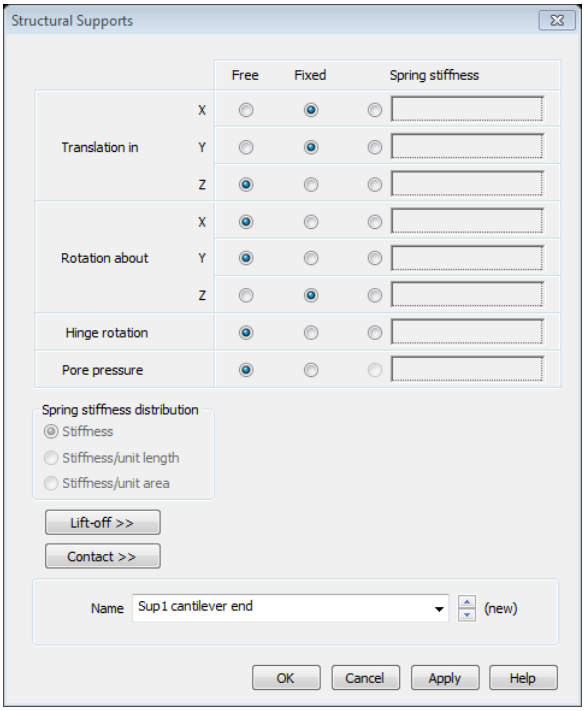
In Lusas we can import a file from Autocad.

- **How to draw in Lusas:**
- Start from 2 points and a line in between:

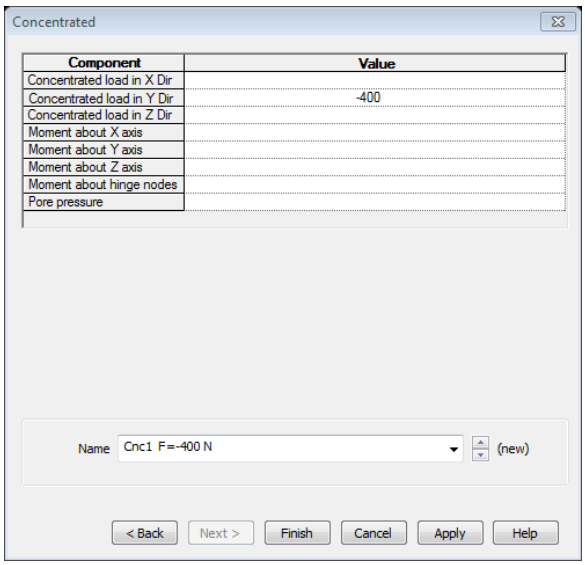
1) Introduce coordinates of points (*new point*)
   (pay attention to units that you choose from the beggining!)
2) Select the 2 points and click *line*
3) Mesh the geometry: *attributes*, *mesh line* , ***thin beam***, *2D, interpolation order: quadratic, number of divisions: 10* (10 cm long each finite element, in total there are 11 nodes), *name: thin beam*



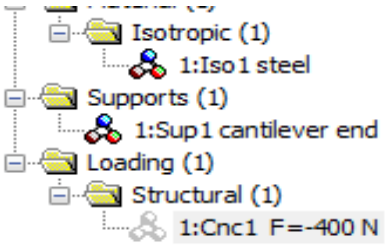
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 7 di 104



7) Loads: *attributes, loading, concentrated, next, y: -400* (because the force is downwards but y axis is upwards)*, apply, finish*



8) After defining the properties we have to select the beam and assign them to it: *left click*, *assign*; (the three cubes color).



!!! The supports must be assigned to the points (not to line in this case)

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 9 di 104

- **Calculation of stress in a given rectangular cross section**

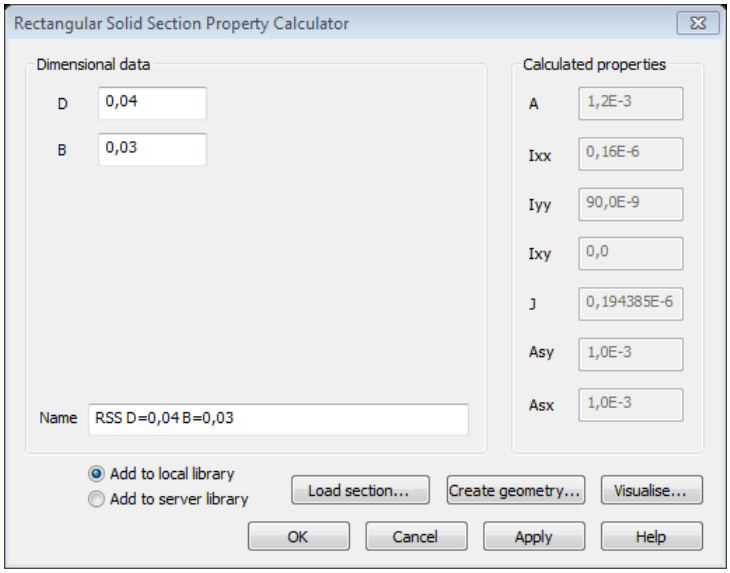
- All assigned attributes are taken away from the thin beam (but the attributes remain in the memory): *Attributes, deassign, from all.*
- Create a new **Thick beam** ("trave spessa"): *attributes, mesh, line, thick beam, 2 dimensional, linear interpolation, number of subdivisions 10, apply, ok.*

!!! If you increase the number of elements, there is a value after which the solution doesn't change visibly, in fact the results will differ only for negligible centesimal numbers.

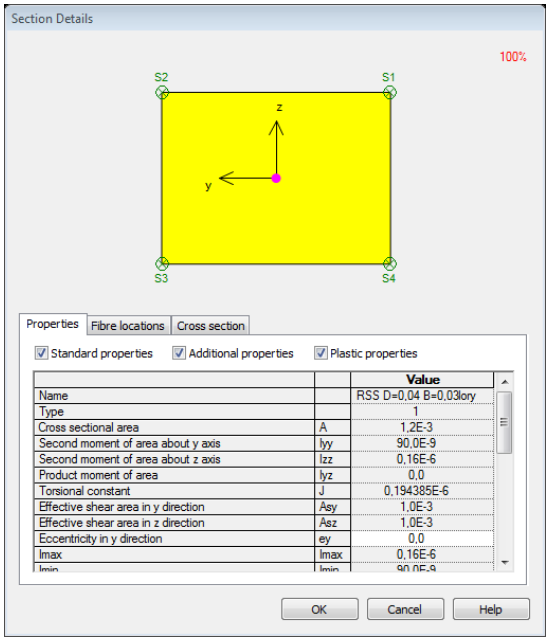
- Assign all properties to the thick beam: *right click, assign to all.*
  Now all the cube-triangles near the thick beam (in the attributes window) are colored.
- Change the line attribute: *attributes, line, 2D thick beam.*

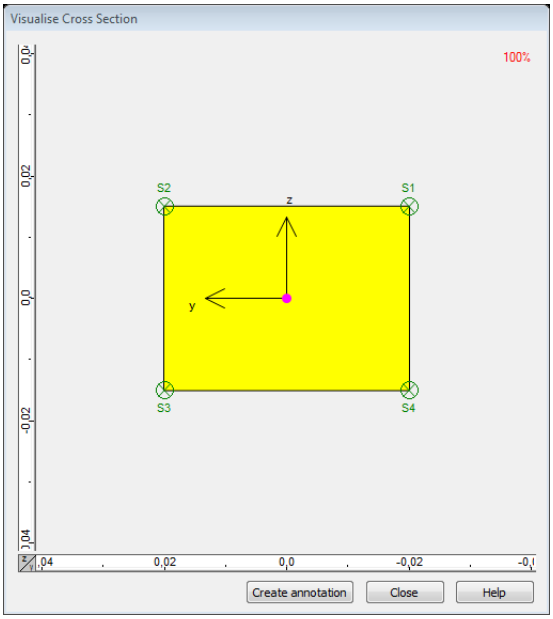
!!! "Prima di assegnare il tipo vincolo nodale e la forza concentrata bisogna selezionare il nodo sul quale si vuole agire, altrimenti ciò verrà applicato a tutti gli 11 nodi" – Before defining the constraints and applying concentrated forces, you have to select the node , otherwise they will be applied automatically on all the 11 nodes of the structure.

- [To not visualize the deformed structure: *right click, deformed mesh* (it must be deselected).]
- Create cross section: *Utilities, sections property calculator, rectangular, solid, add to local library* (the Excel file will be saved in the current folder and it can be used only there (server library is for all folders)*, apply, ok.*



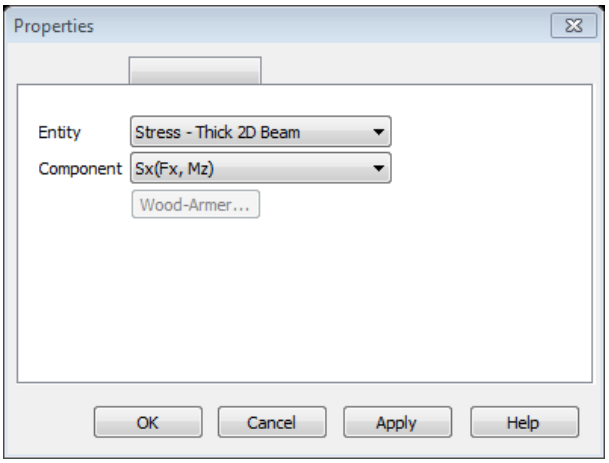
- Assign the section created to the thick beam: *Geometric line, right click on thick beam, definition from library, user sections, (*you can choose the rotation of the angle: *rotation about centroid 0°).*





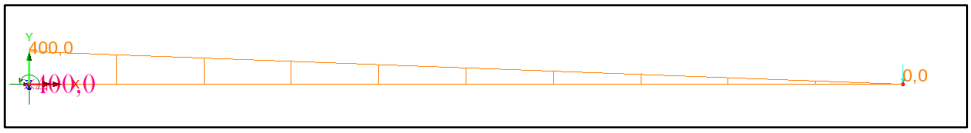
RUN!!!

- **How to see different diagrams:**
  you can see the different diagrams and values of stress, reactions (M,N,T) by changing the properties: *layers, right click on diagram, entity: …. , component: …. .*



- Change font, colors, dimensions: *right click on the elements in the main window, font.*

Diagram Mz and reaction Fy:



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 13 di 104

-activate/deactivate the flashing and change vista to see in 3D and turn to the 2D

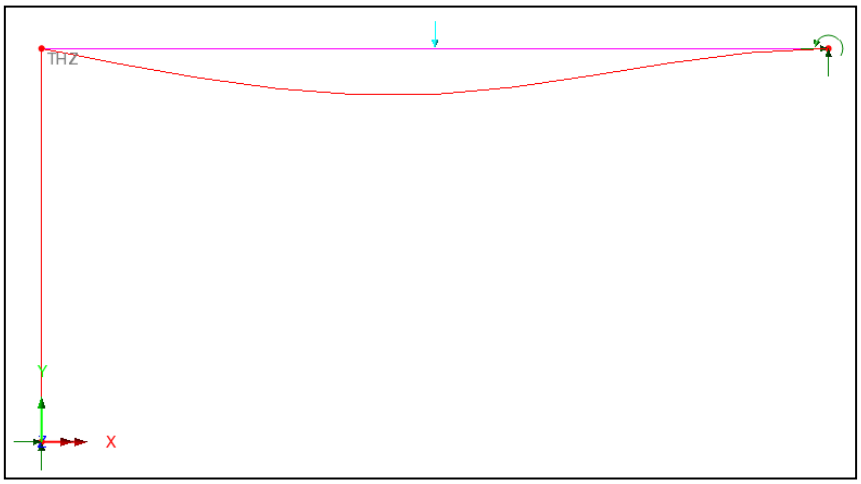
- ***deassign from selection the mesh, click on end conditions…: start of line: fixed; end of line: rotation around z:pinned*** (*end conditions…* is available only for thick beams)

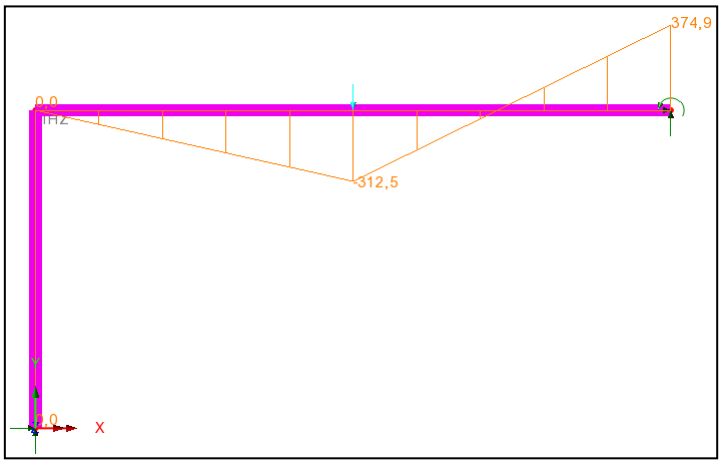
-material, isotropic

-support ,one is pinned and one is fixed (node 1 and node 3)

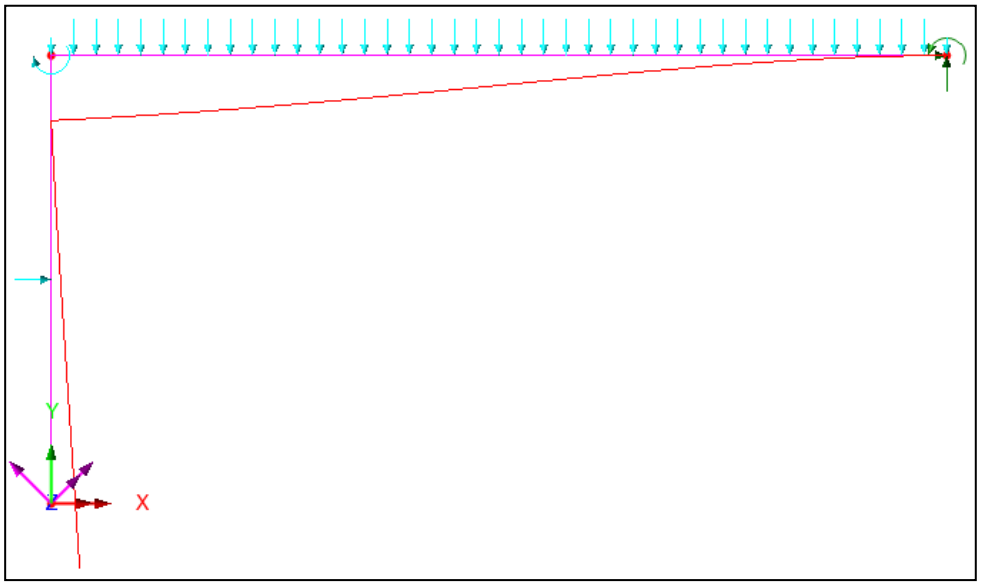
-loading, internal beam point, next, global load direction, load position about beam axis, distance type parametric , so you can write L/2=0.5; PX,PY are components*: Py=-1000, name, apply, finish*

RUN!!!

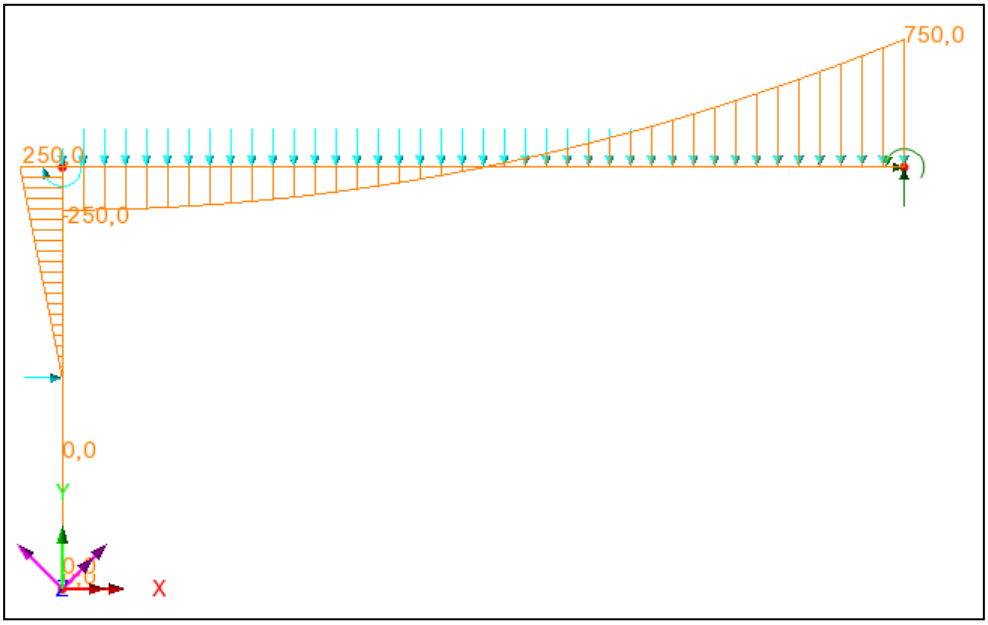




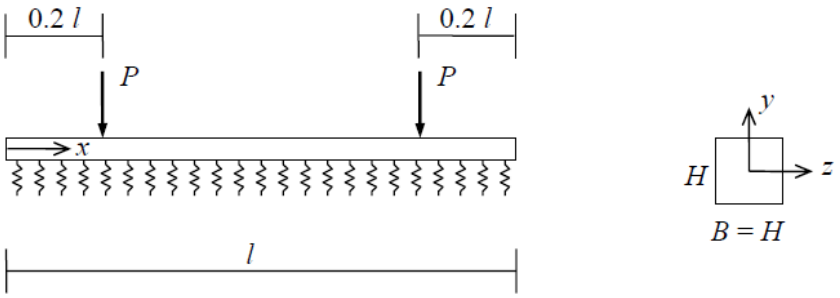
- Deformed mesh:



- Moment Mz diagram:



Beam on elastic foundation – Winkler



Data:
$l = 10$ m
$B = H = 0.4$ m ($A = 0.16$ m^2; $I_z = 2.13$E-3 m^4)
$P = 1$E5 N
$E = 30$E9 Pa, $\nu = 0.18$ (concrete)
$K = 4$E7 Pa (soil stiffness) => Spring stiffness distribution in LUSAS: use "Stiffness/unit length'

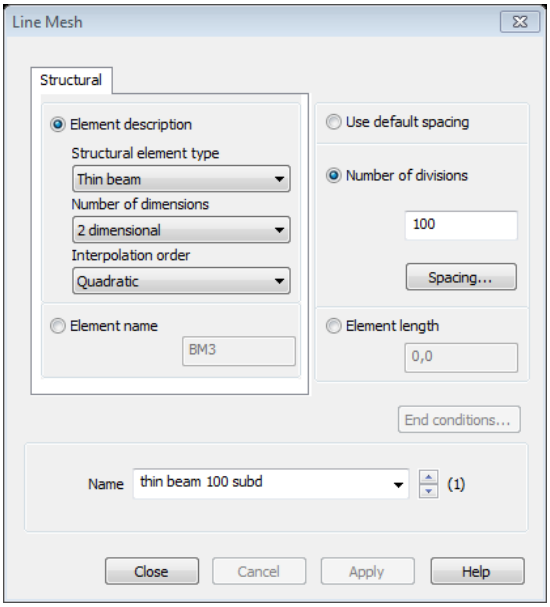
Extreme cases:

1st case: $K = 4$E10 Pa

2nd case: $K = 4$E3 Pa

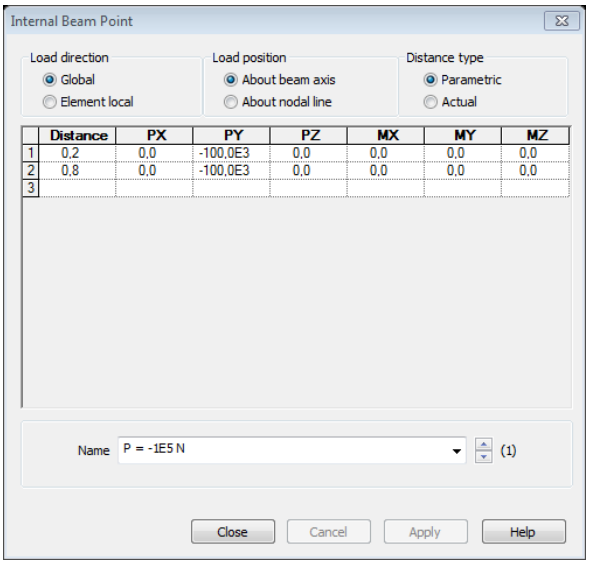
- **How to draw and assign the soil:**

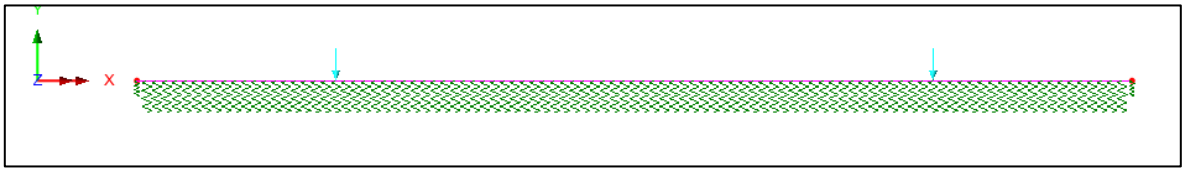
- Draw a line and assign mesh



- Assign geometry and material

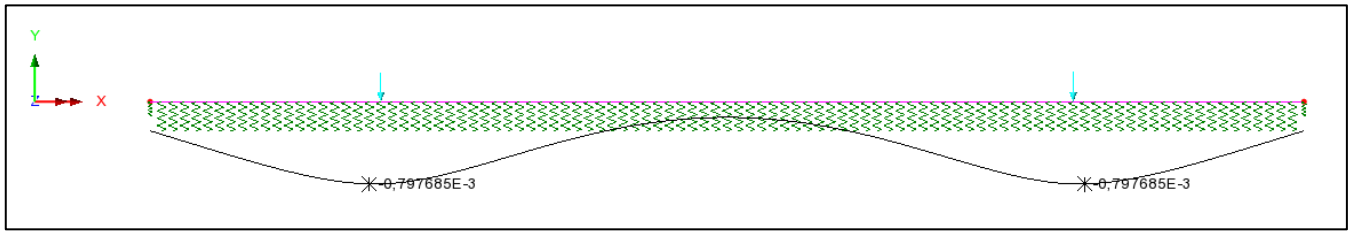
- Assign the loads



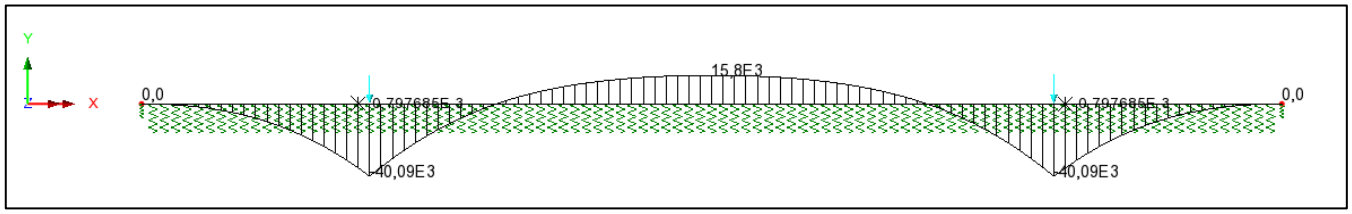


RUN!!!

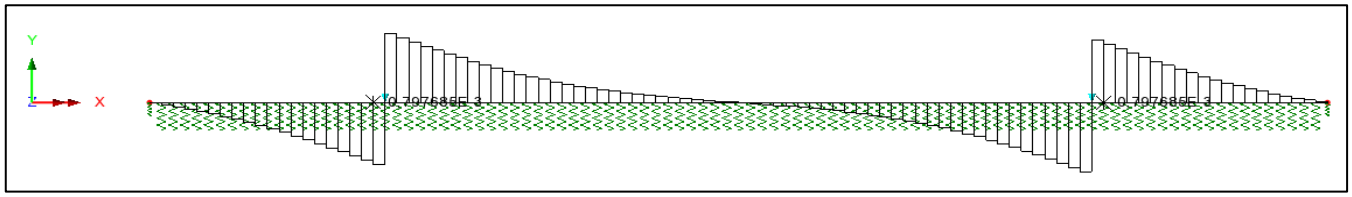
Deformed mesh:

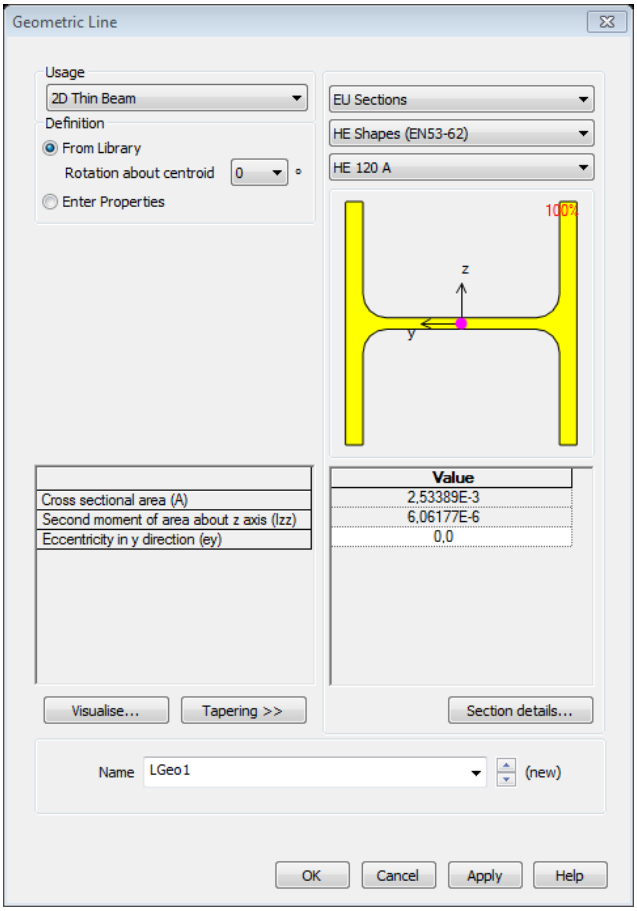


Moment diagram Mz:



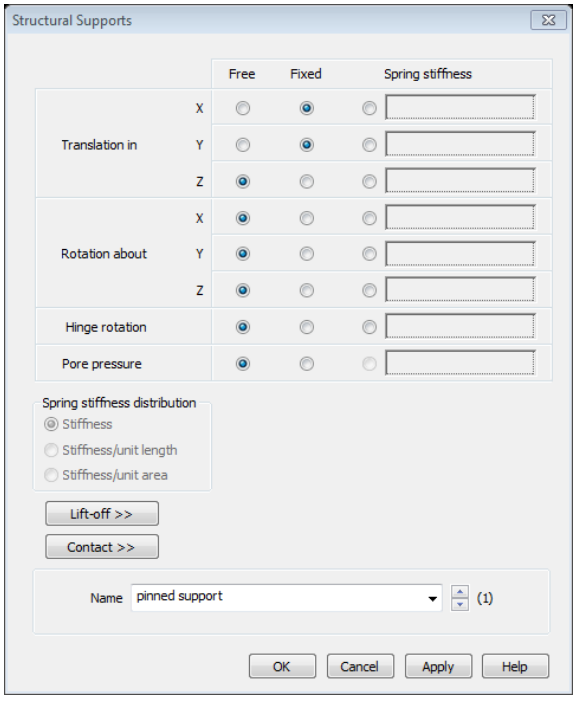
Shear Fy:

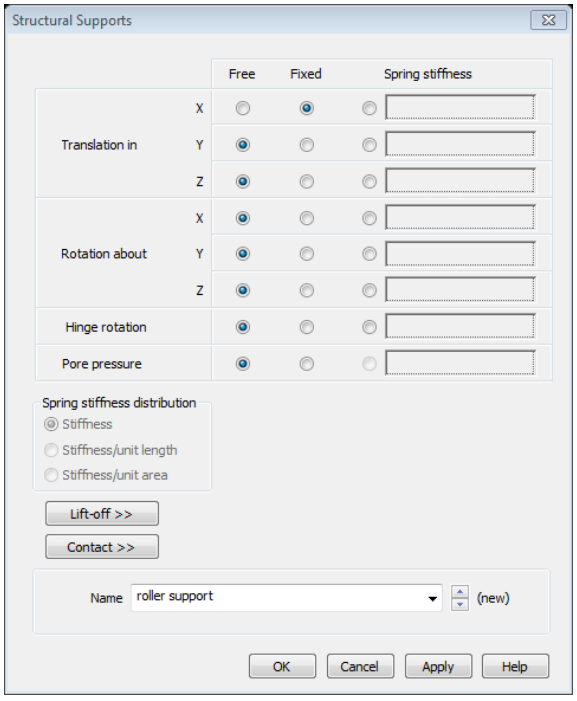




-Attrib, materials, isotropic,…

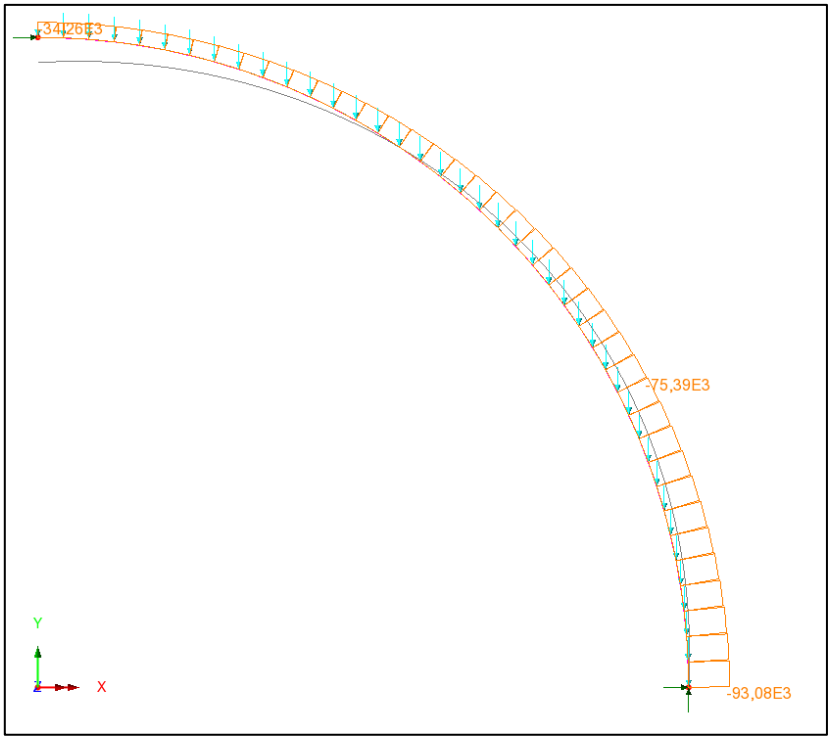
-Attrib, supports,…



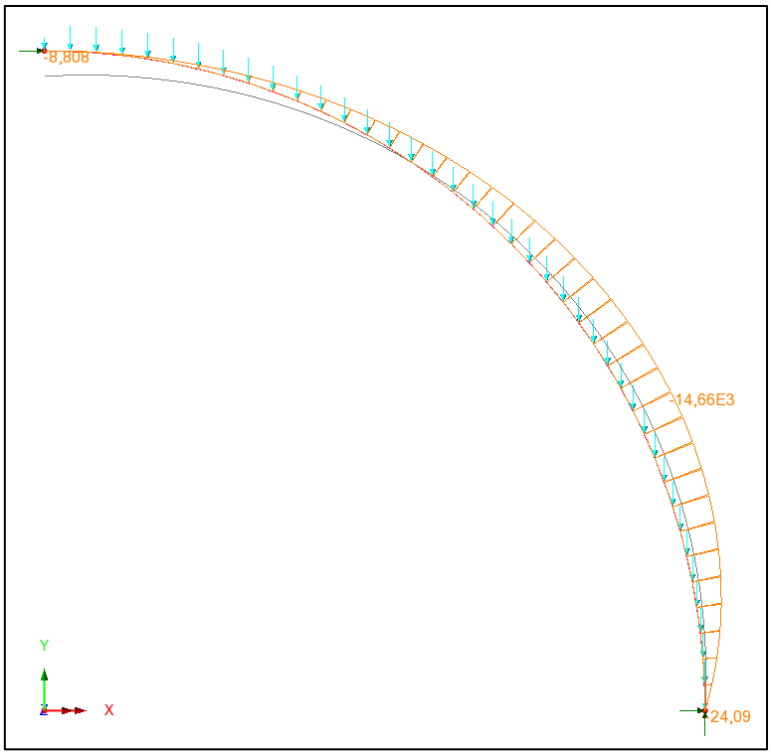


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 23 di 104

Compression Fx:

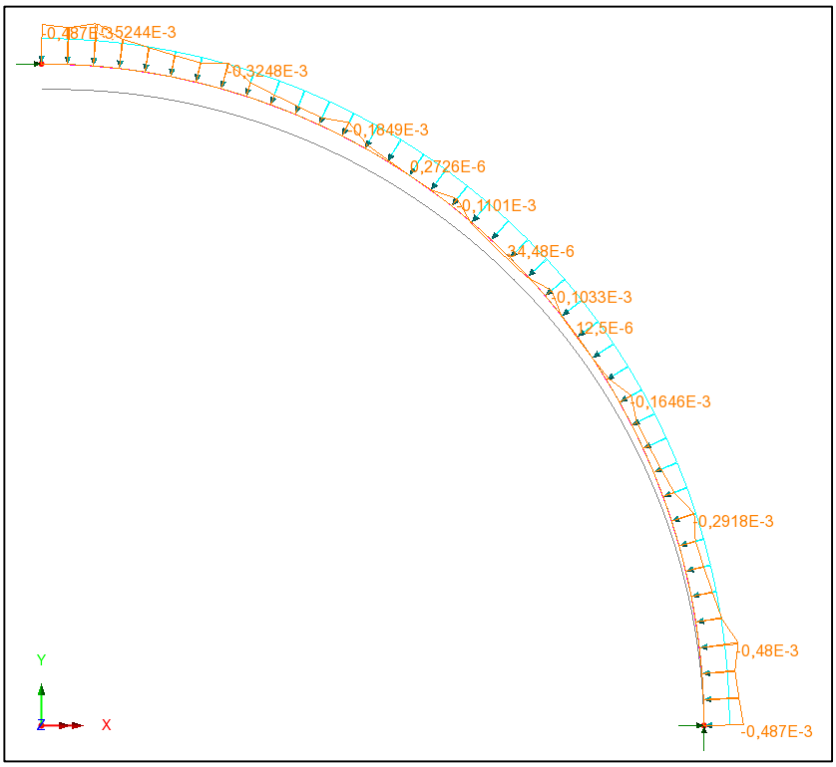


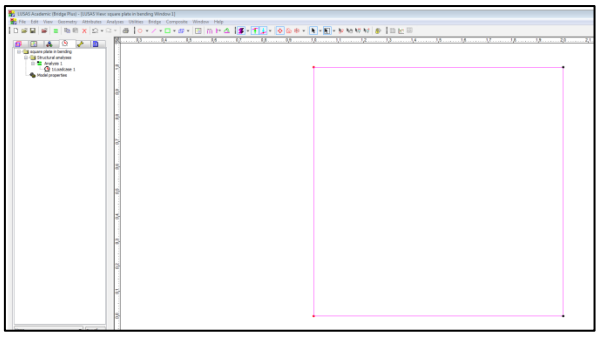
Bending moment Mz:



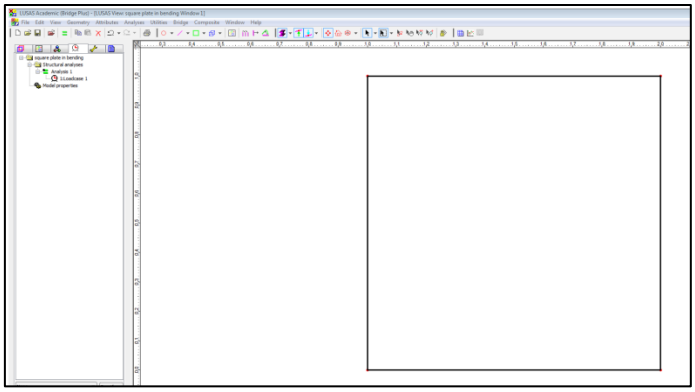
RUN!!!

Moment Mz:

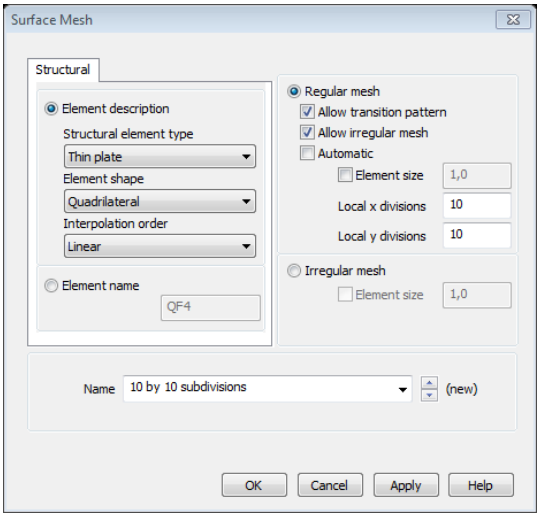




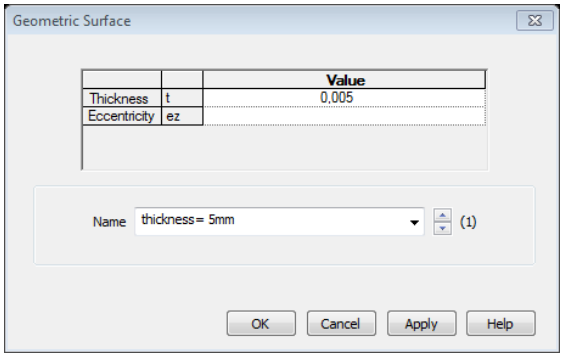
-create a surface: select 2 lines, *geometry, lines, by joining*



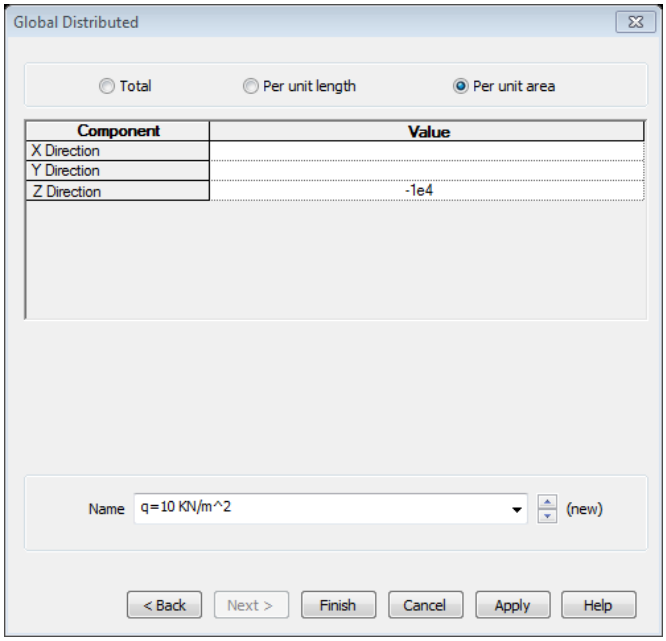
-attributes, mesh, (with regular spacing), surface, thin plate:



-attributes, geometry, surface:

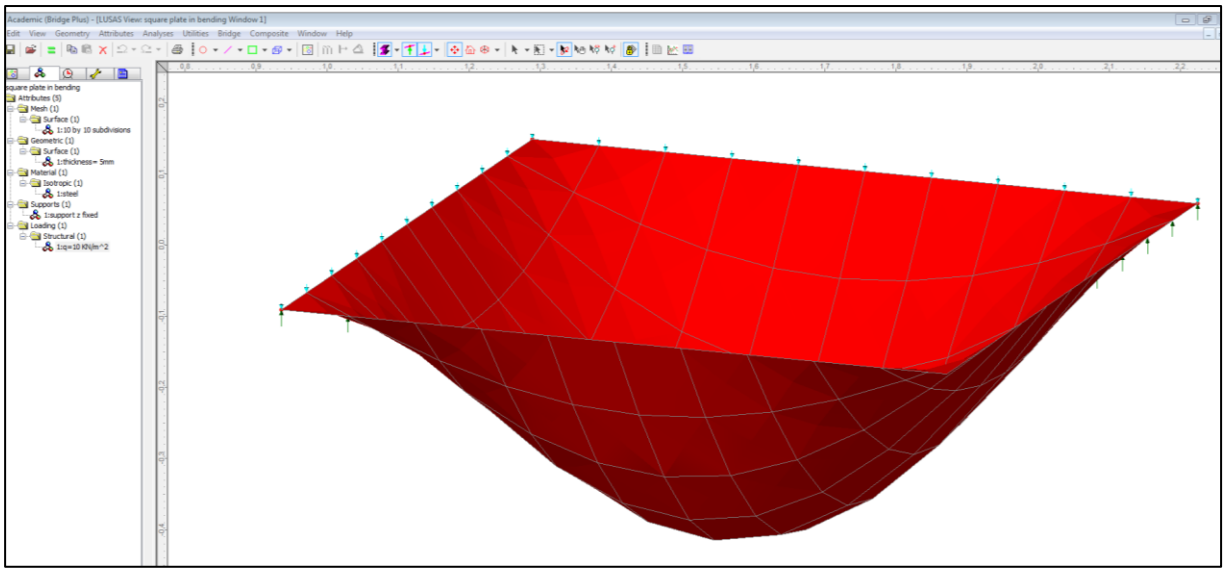


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 29 di 104



RUN!!!

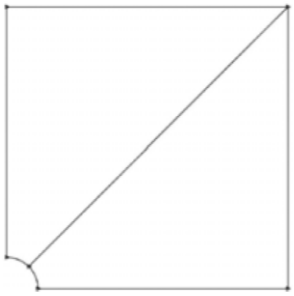
- deformed mesh: magnitude 100 mm, mesh: solid



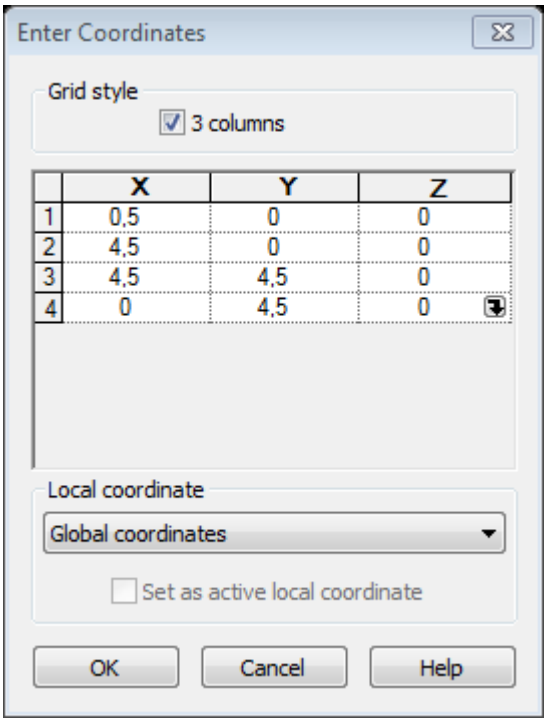
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 31 di 104

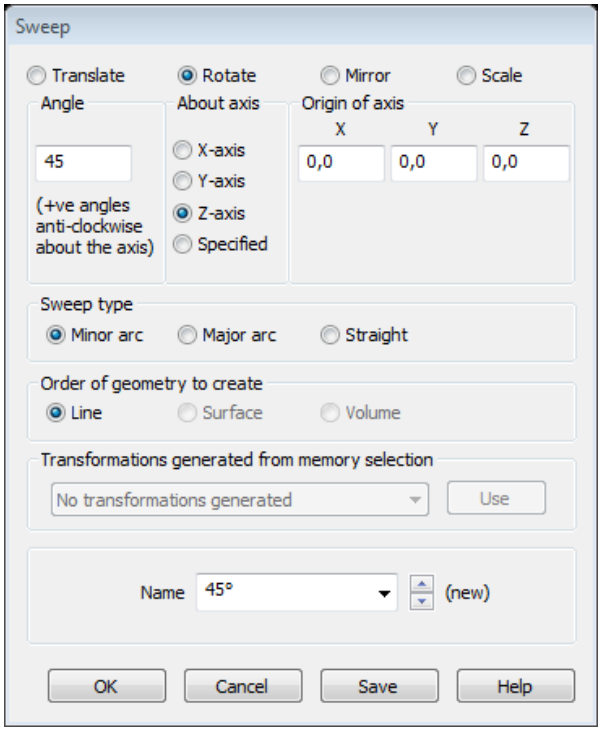
- **How to draw ¼ of plate:**

Exploiting the symmetry of the problem, model one fourth of plate:

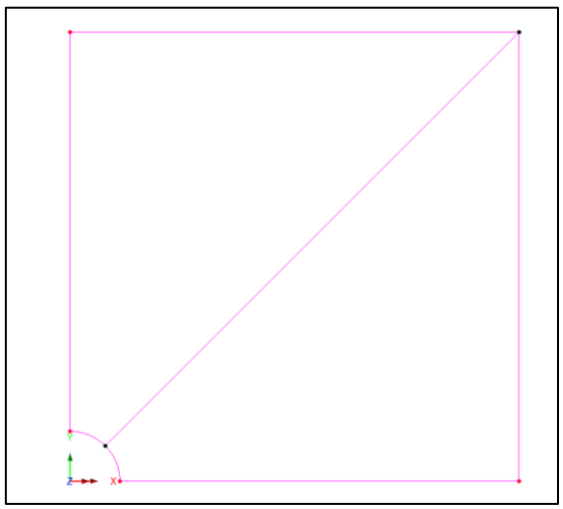


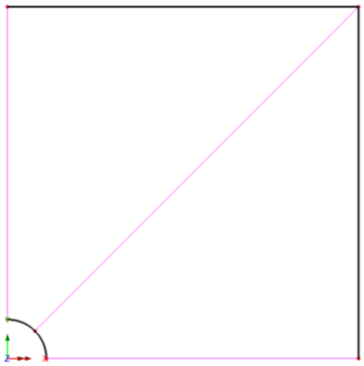
-Insert points: write the coordinates of the corners squared plate and we can find the 3 nodes of the arch by sweeping 2 times the point (0,5; 0): *geometry, line, by sweeping, rotate, 45°, ok*.



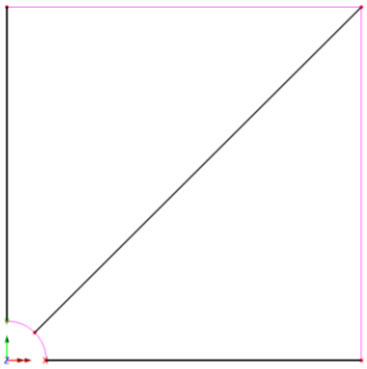


-Connect all points by a line in a counter-clockwise direction: *select 2 nodes, new line*



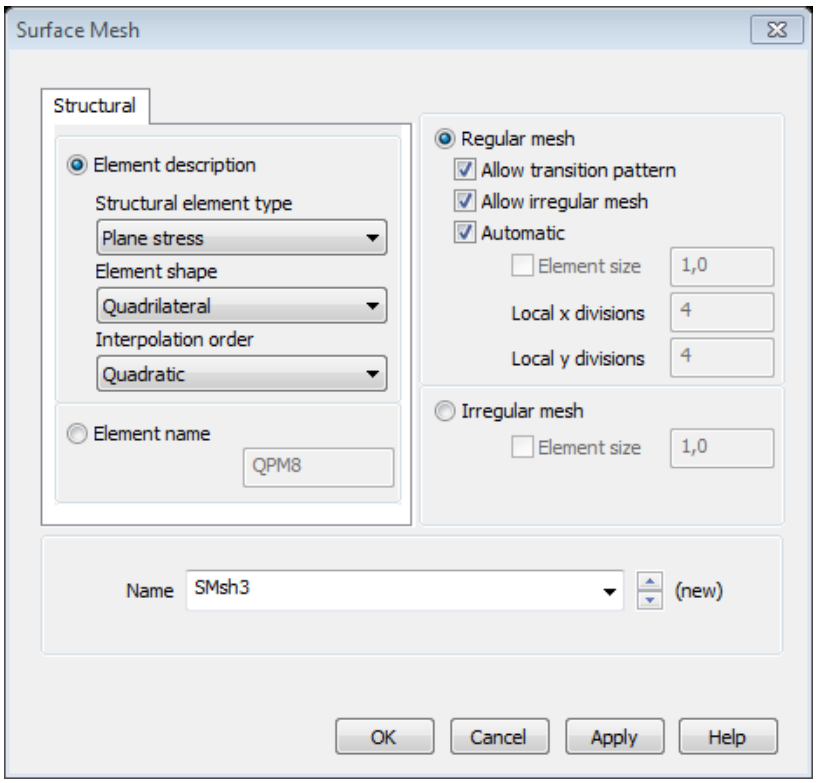


…. And the 9 subdivisions mesh to:

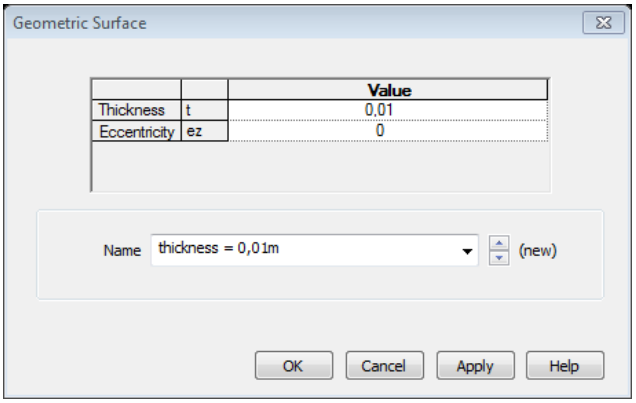


-*Geometry ,surface line,* (create a triangle on the right and one on the left)

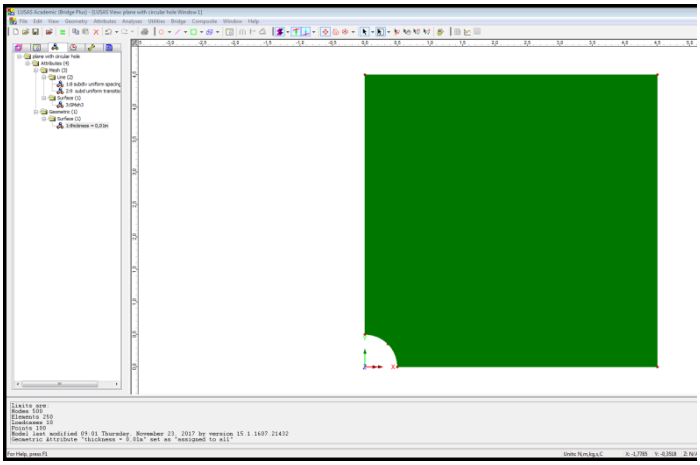
-*Attributes, mesh, surface, plane stress :*



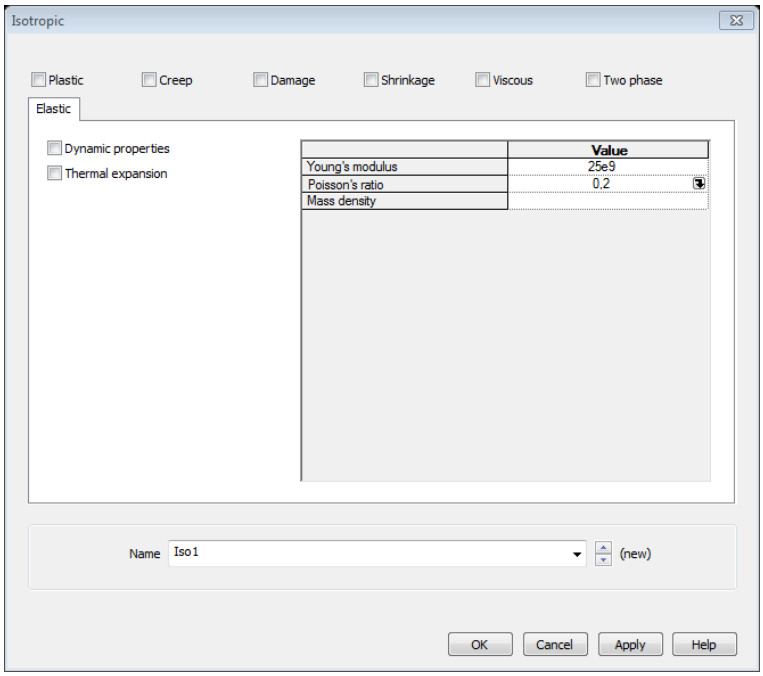
-Attributes, geometric, surface



-Assign it:

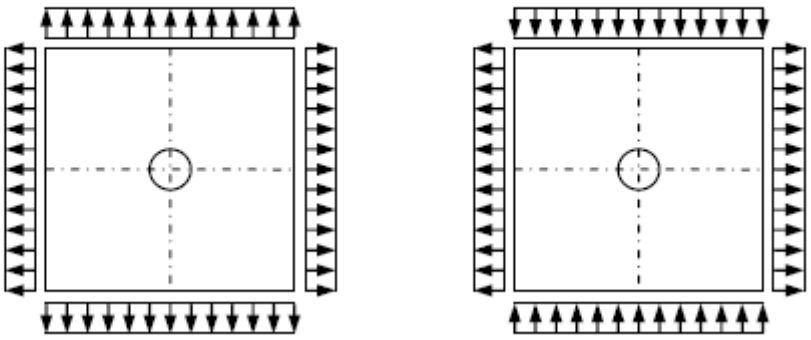


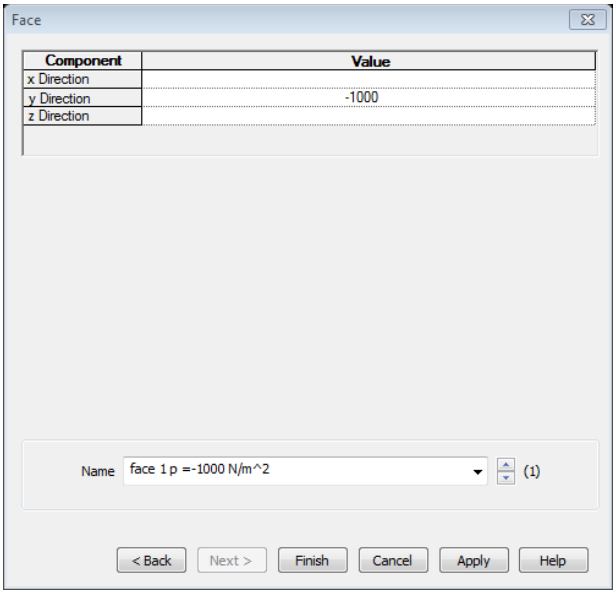
-Attributes , materials ,isotropic



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 37 di 104

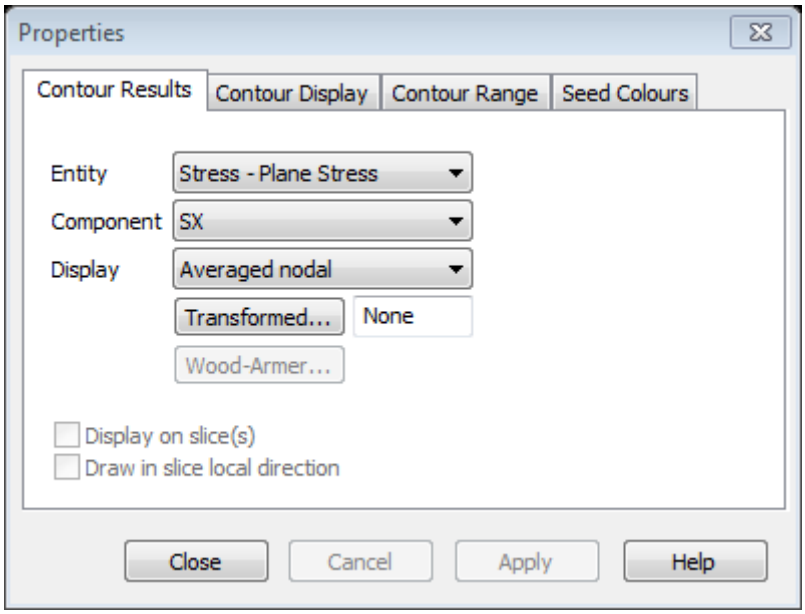
Different loading conditions:

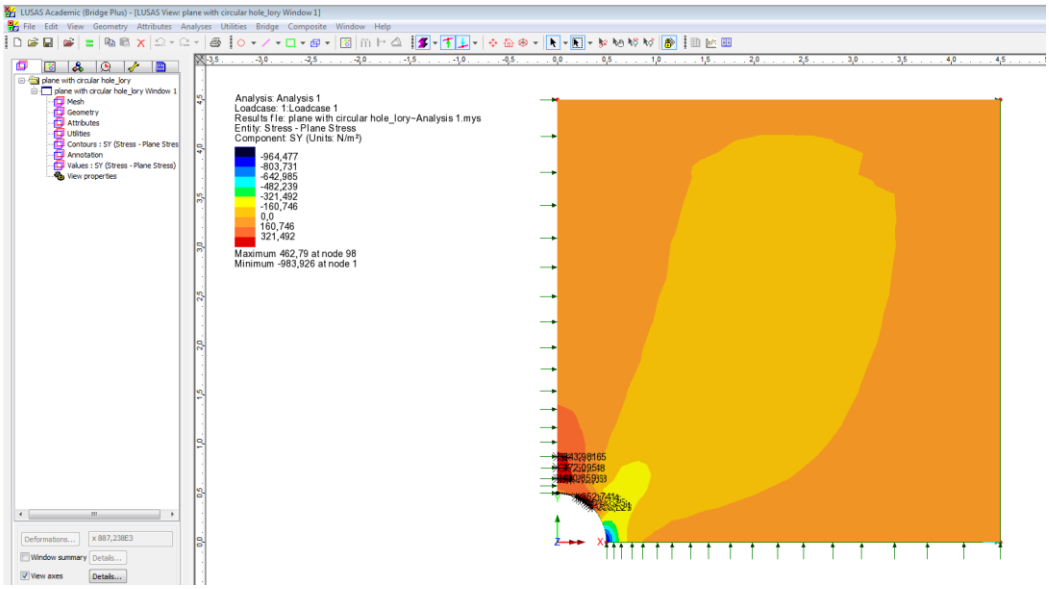




RUN!!!

- *Right click, contours, entity plane stress, **SX***

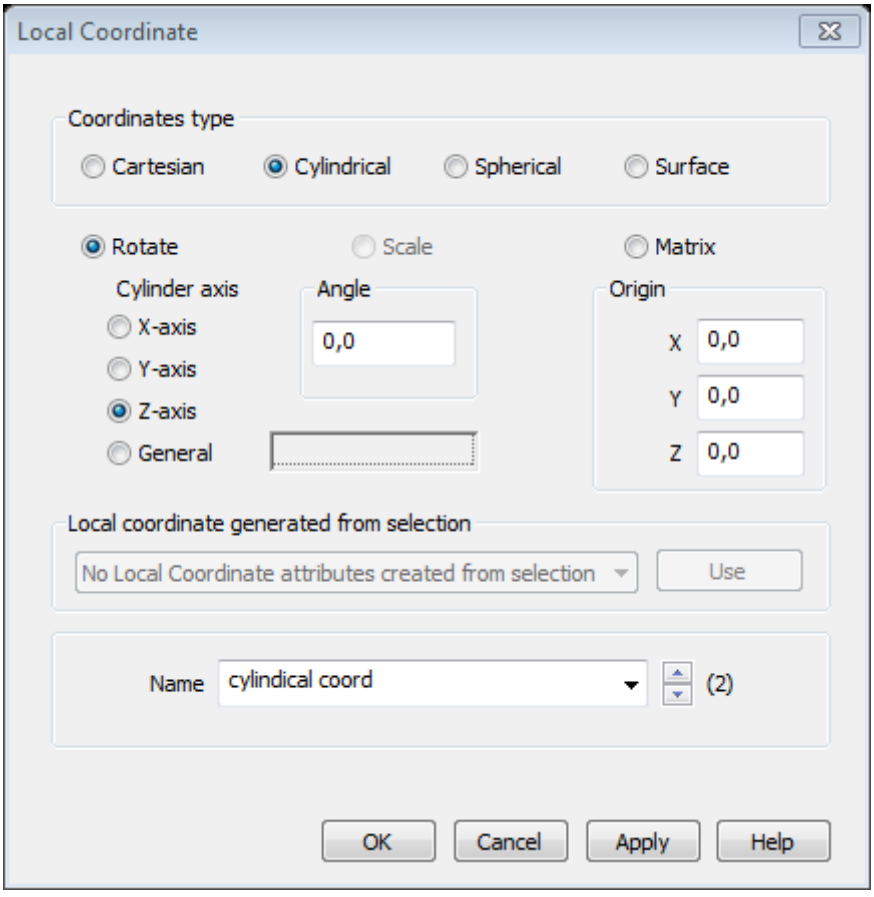




- **How to change RS from Cartesian to cylindrical:**

It changes Radial distances \ Angles \ Z vertical.

-Attributes, local coordinates, cylindrical,ok (!!!also used for rotations of 45°)

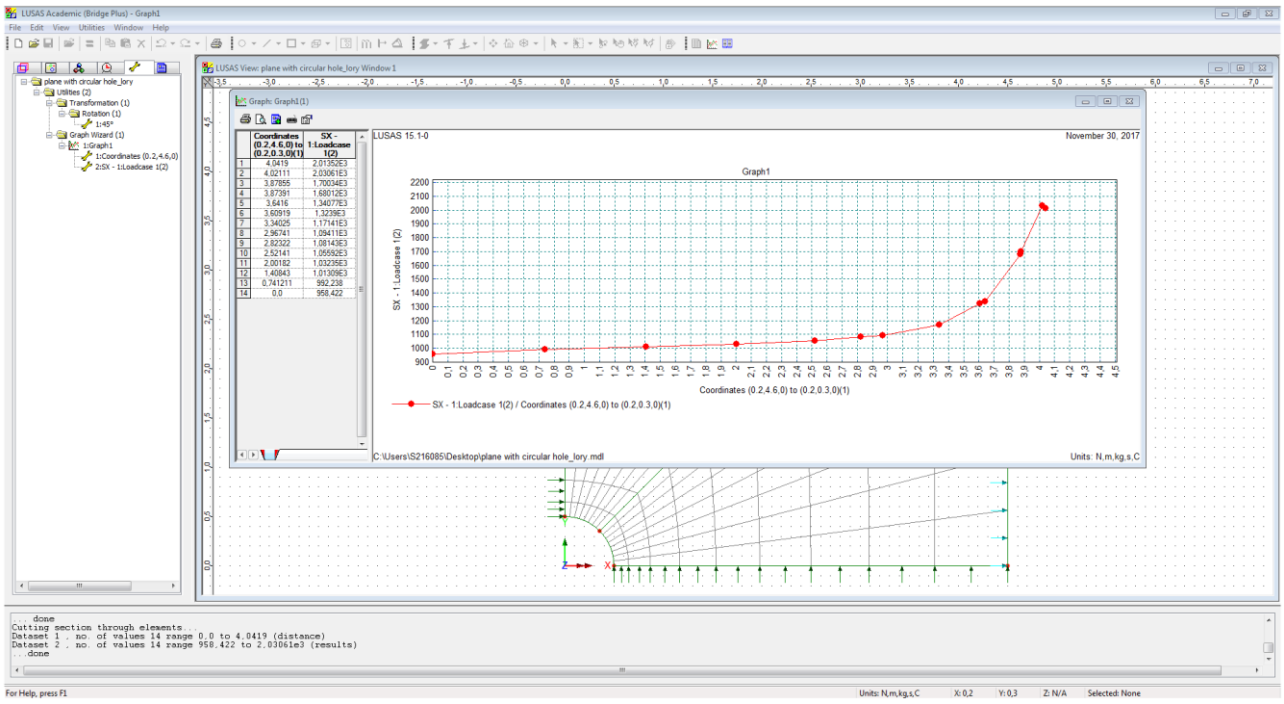


-Assign to all after selecting.

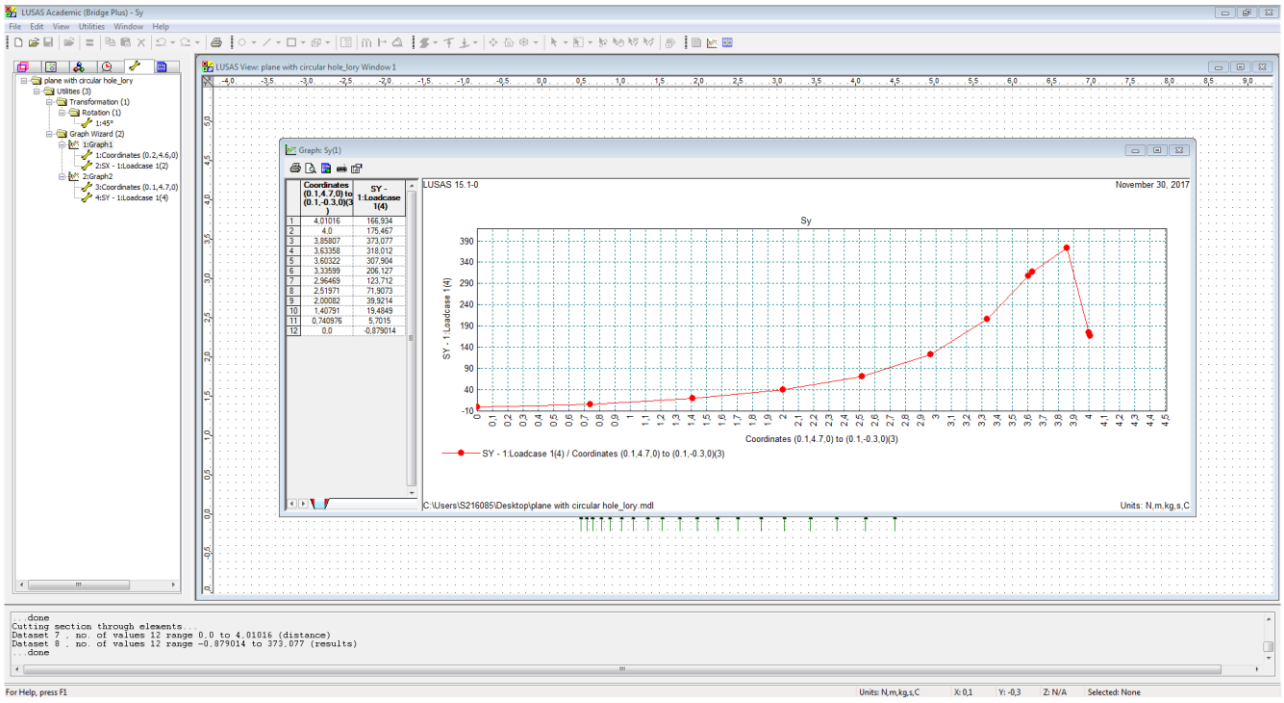
-Contours ,stress-plane, stress

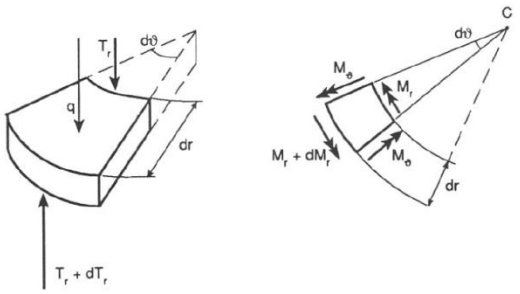
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 41 di 104

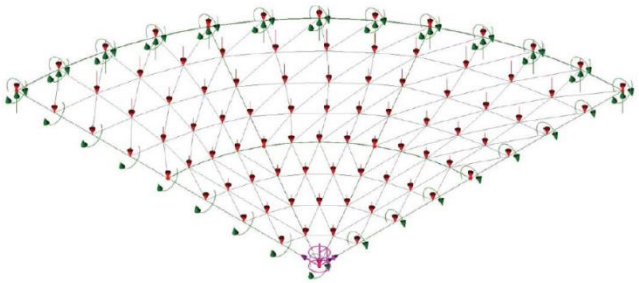
- **For SX:**



- **For SY:**



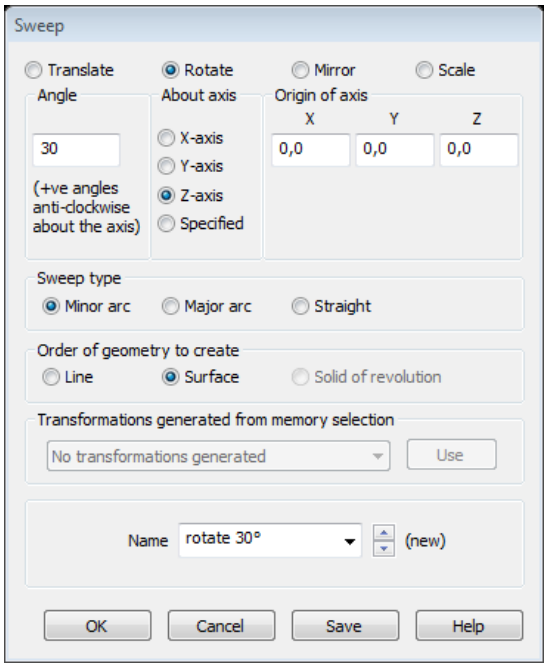




- **How to draw and apply attributes:**

- Do points (0;0), (0.5;0),(1;0) then create 2 line then sweep 3 times the line: by rotating by 30°:

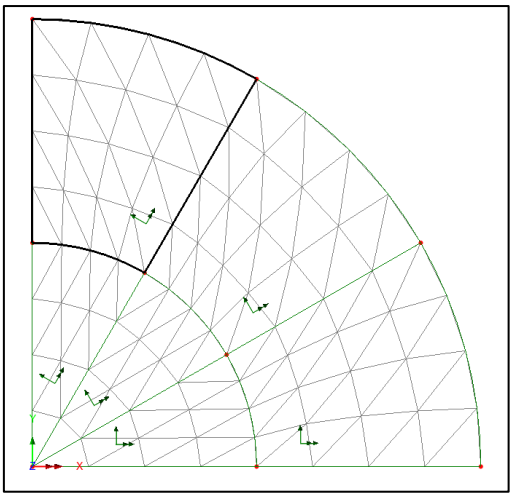
- *Geometry, surface, by sweeping ,*

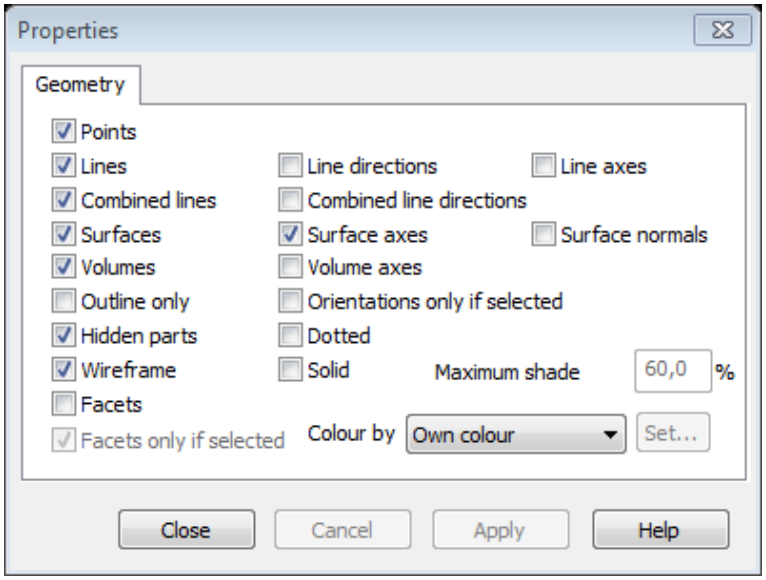


We have created 2 surfaces by sweeping 2 lines.

- Now select the 2 new lines and click : *Geometry, surface, by sweeping* , and select the created “rotate 30°”.

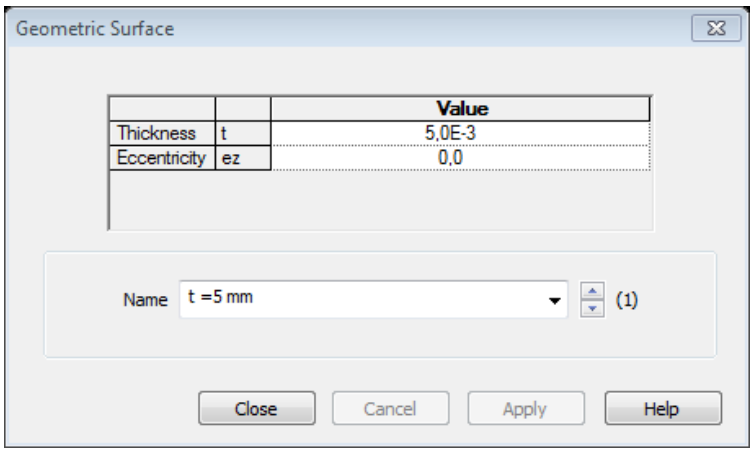
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 45 di 104



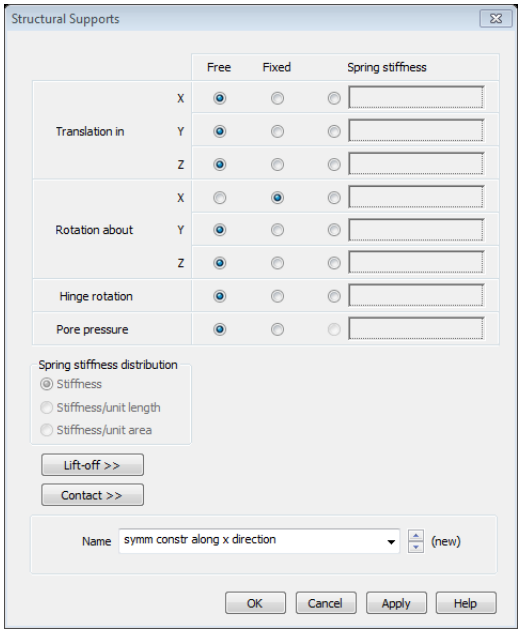


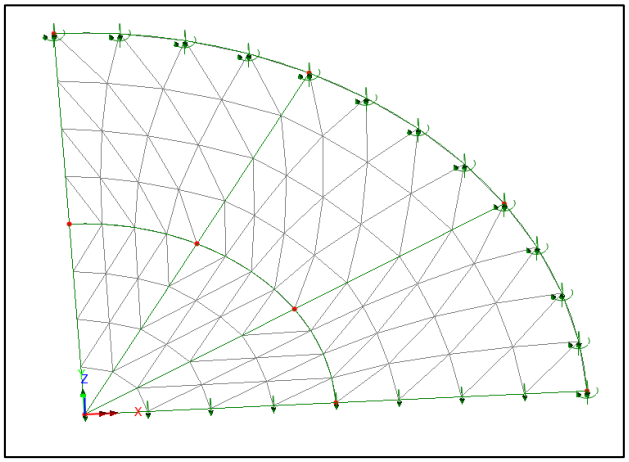
The mesh is not symmetric (no problem).

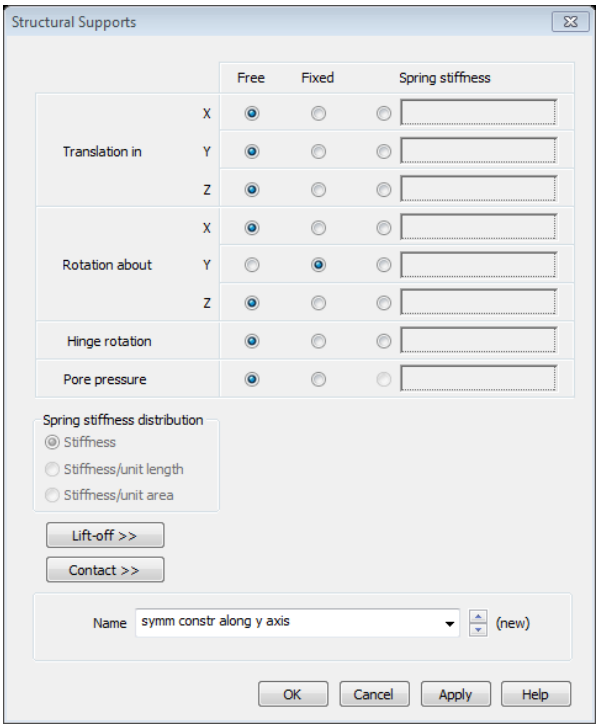
-attributes, geometric,surface, assign thickness



-attributes, material,isotropic,



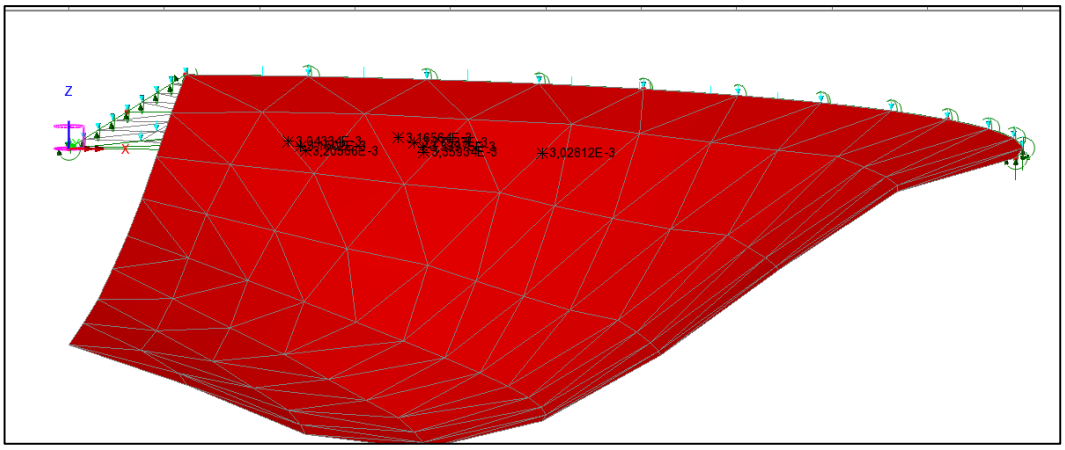


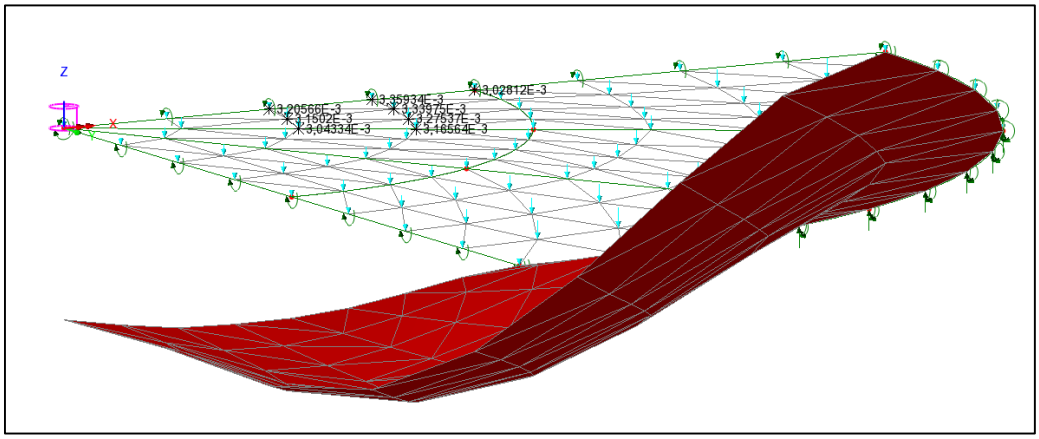


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 49 di 104

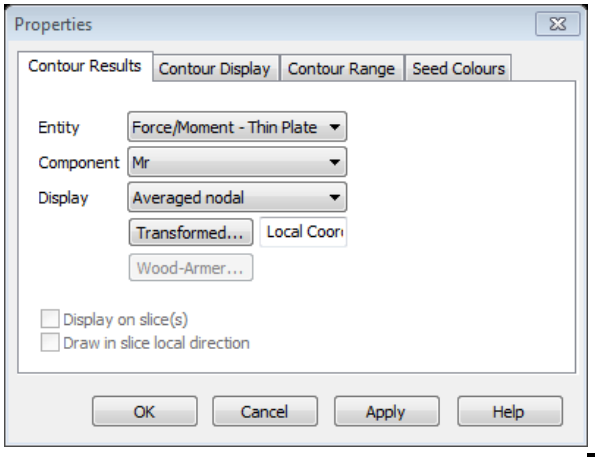
RUN!!!

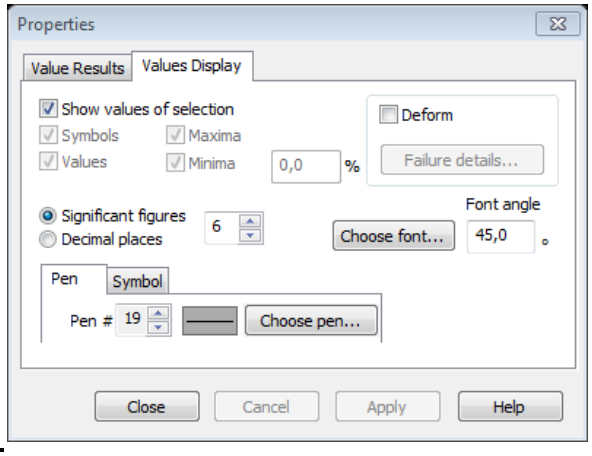
- Deformed mesh:

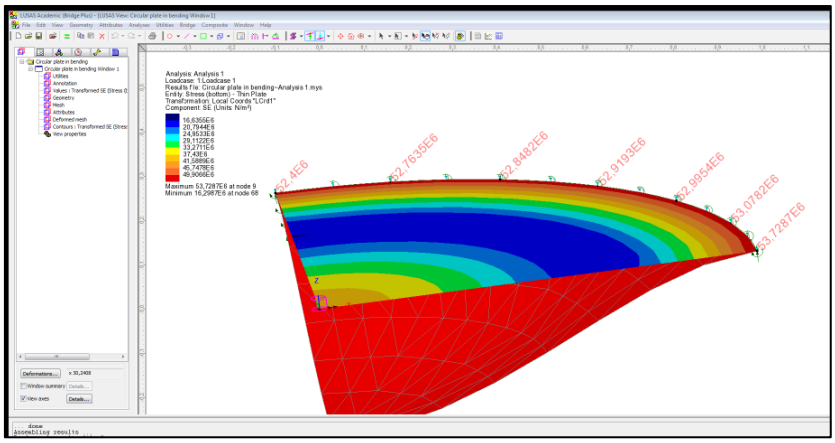




- **Mr component:**

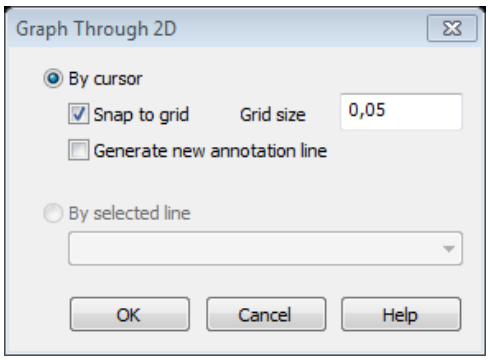


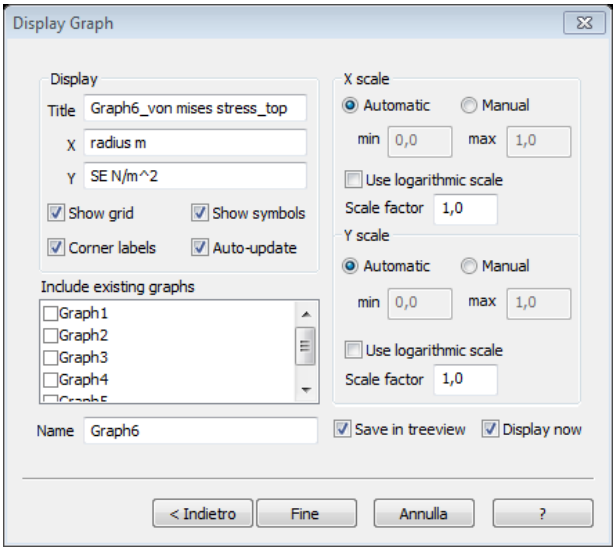


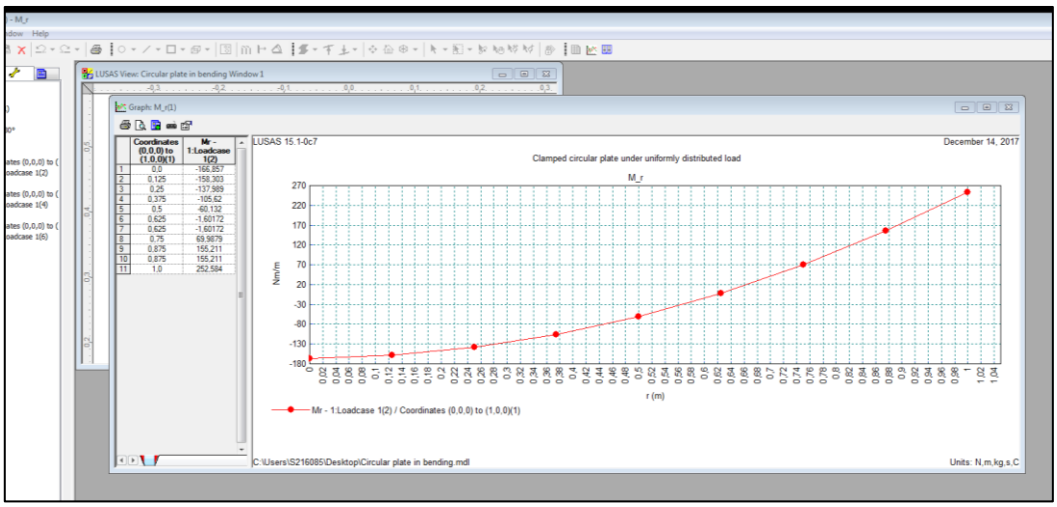


- **How to draw diagrams:**

- Utilities, graph through 2D, grid = 0.05



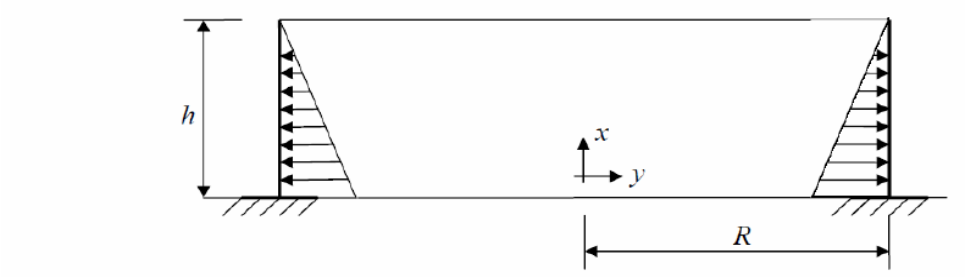




- to see the diagrams, click on utilities (in the right panel-yellow key).

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 53 di 104

Cylindrical tank under hydrostatic pressure



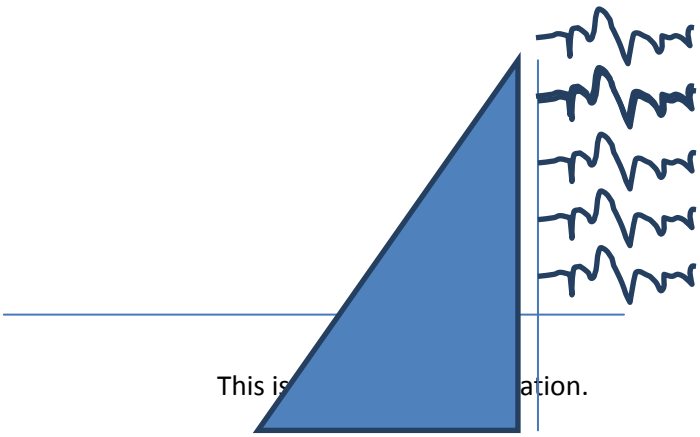
Vertical section

$R = 3$ m, $h = 1$ m, thickness $t = 4E\text{-}4$ m

Steel with: $E = 2.1E11$ Pa, $\nu = 0.3$

Water: $\gamma = 9.81E3$ N/m^3

1. Model one fourth of tank using "Thin Shell" elements (2D).
2. Model the tank by a line using "Axisymmetric Thin Shell" elements.

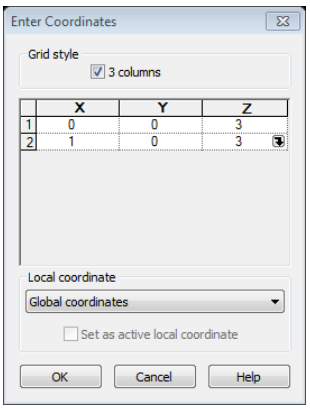


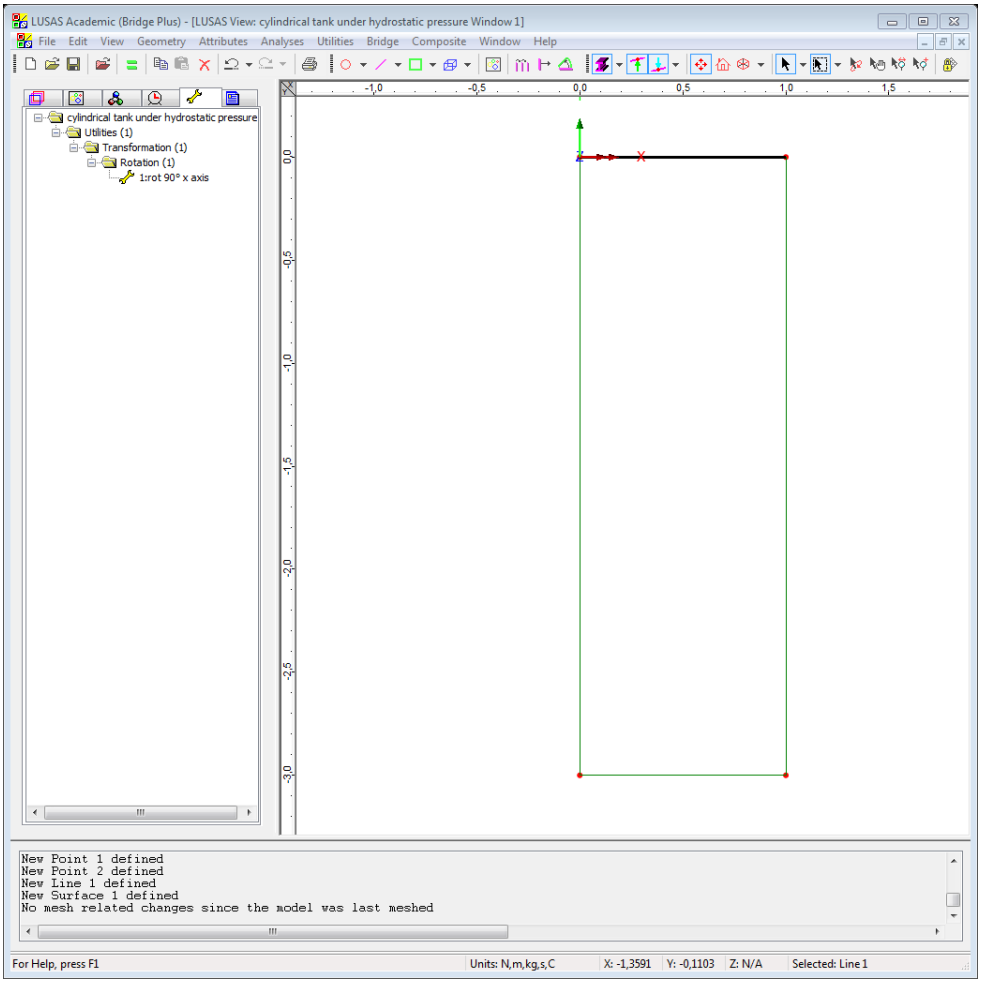
This is ation.

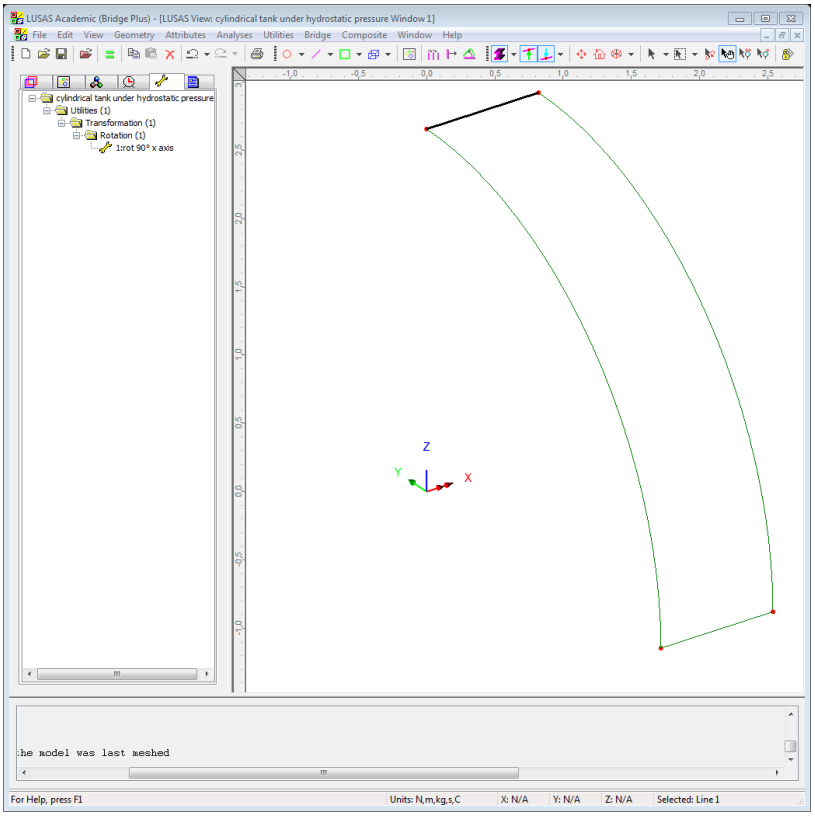
Then we can use the thin shell elements or the axis-symmetric(symmetry around vertical axis)

- **How to draw the tank:**

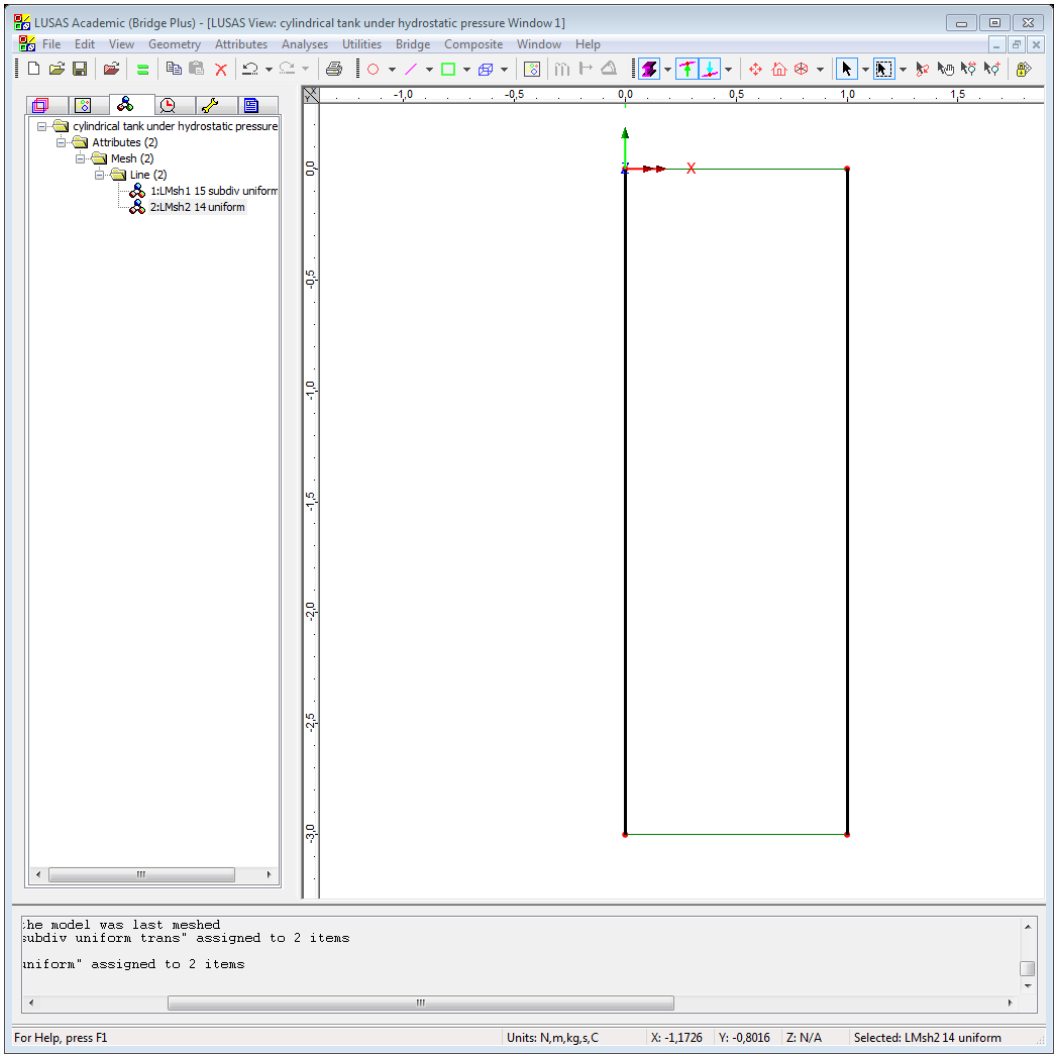
- New point: (0,0,3);(1,0,3) and connect by a line.



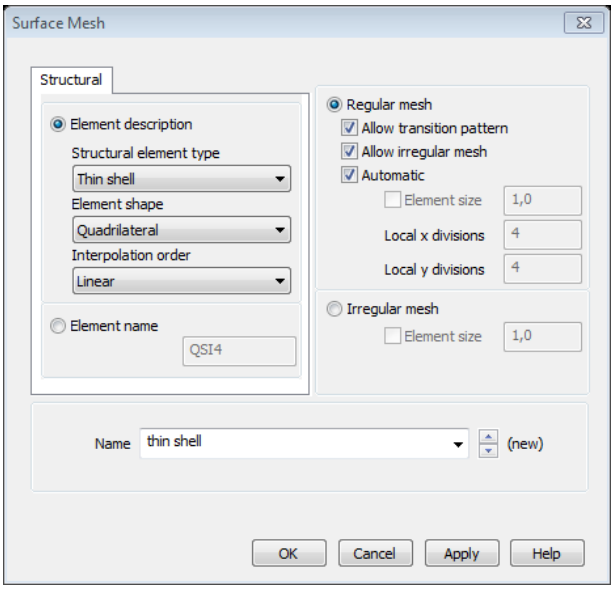




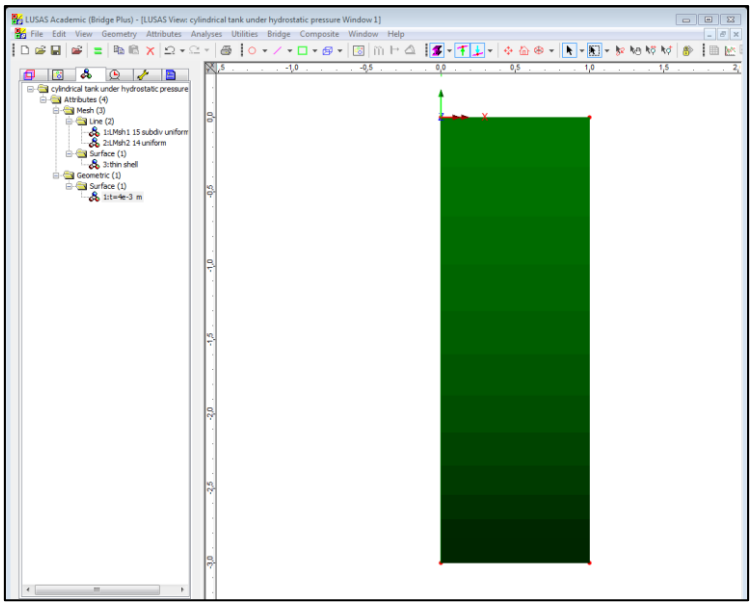
- Assign to the short side the uniform transition and to the circular (long) side the uniform spacing:



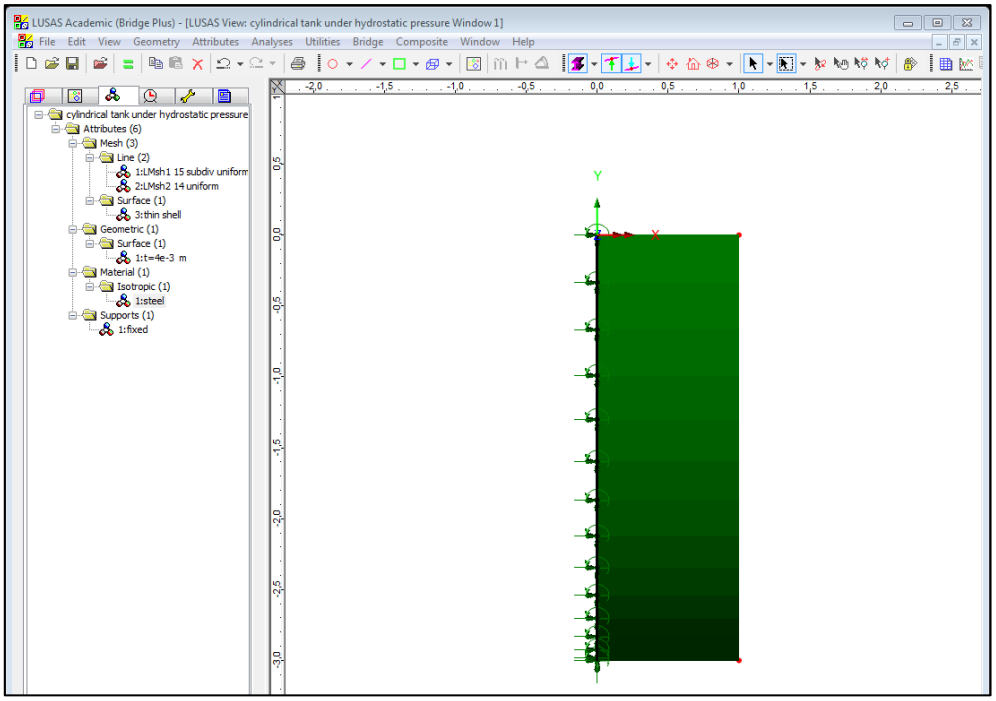
- *Attrib,mesh, surface,thin shell* (Kirchoff theory) = no shear deformation:



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 59 di 104

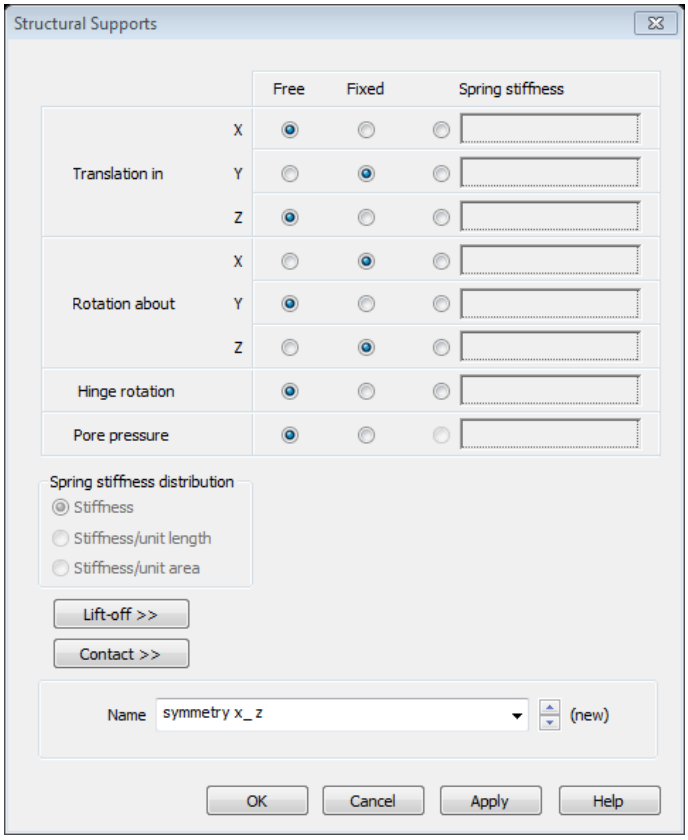


- *attributes, supports,…*

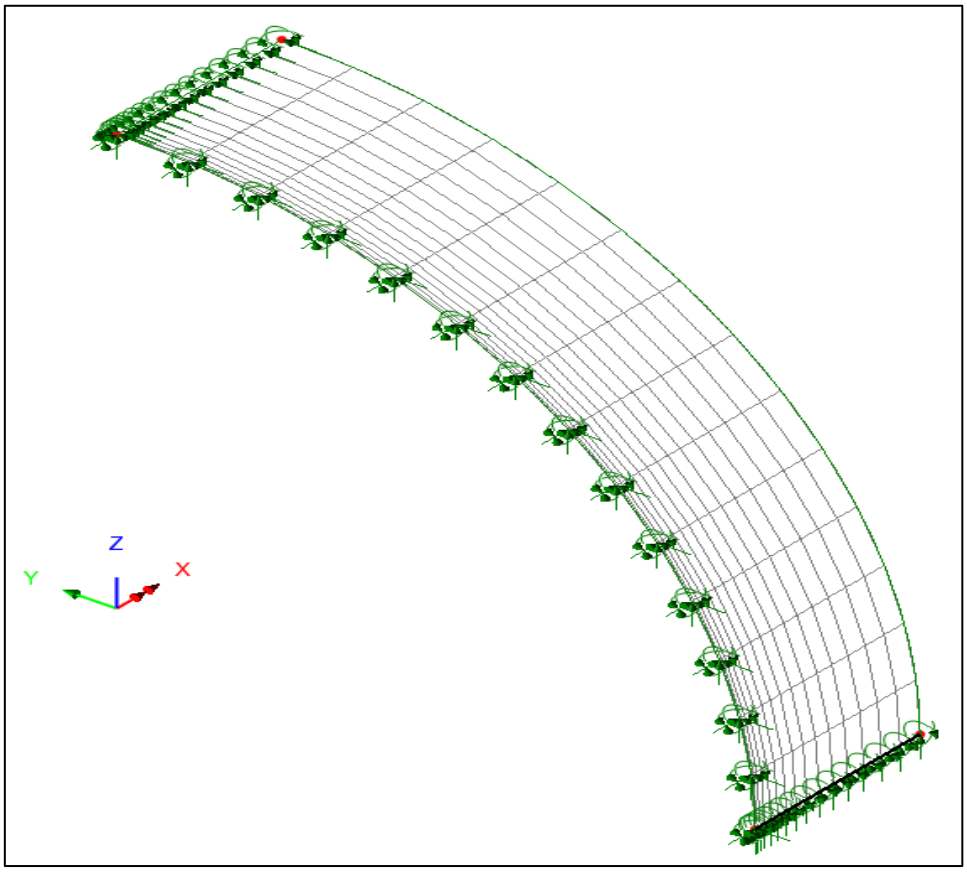


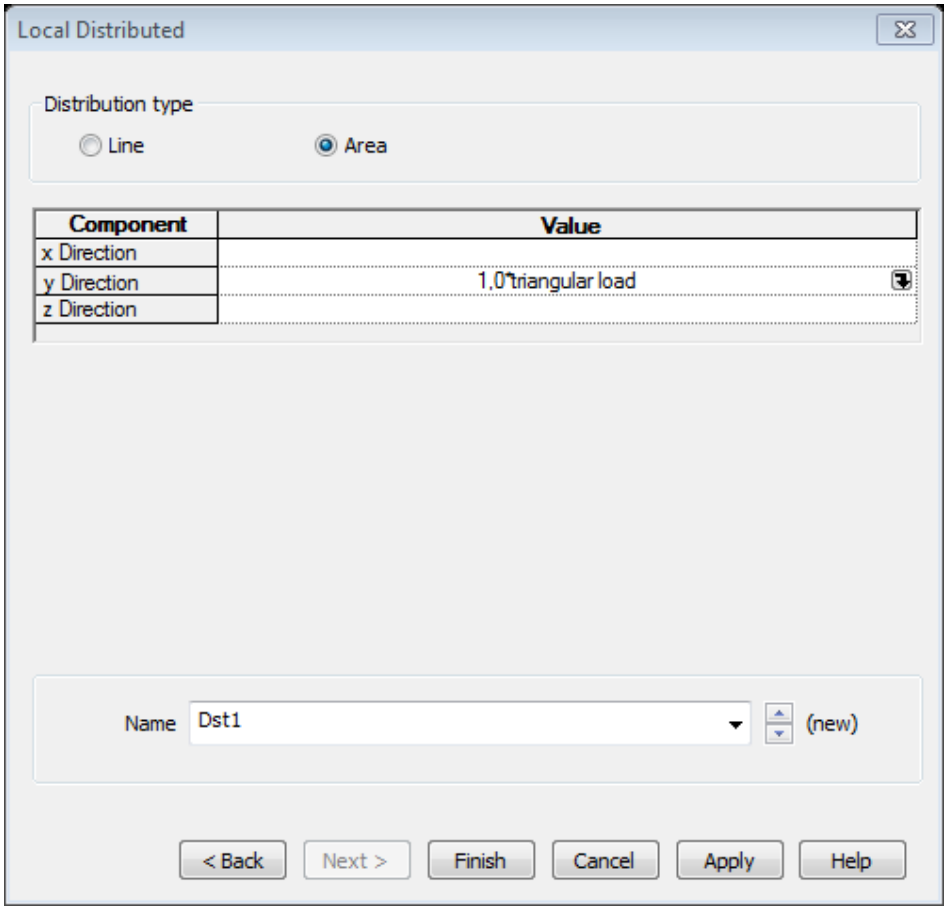
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 61 di 104

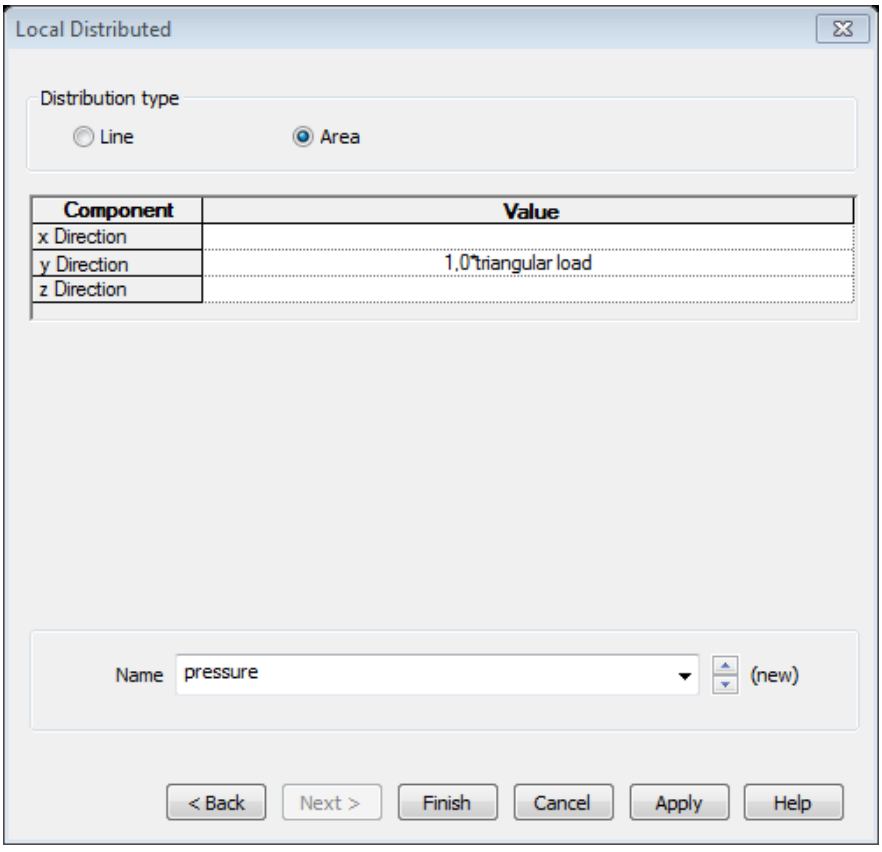
- Symmetry x-z: *attributes, supports,..*



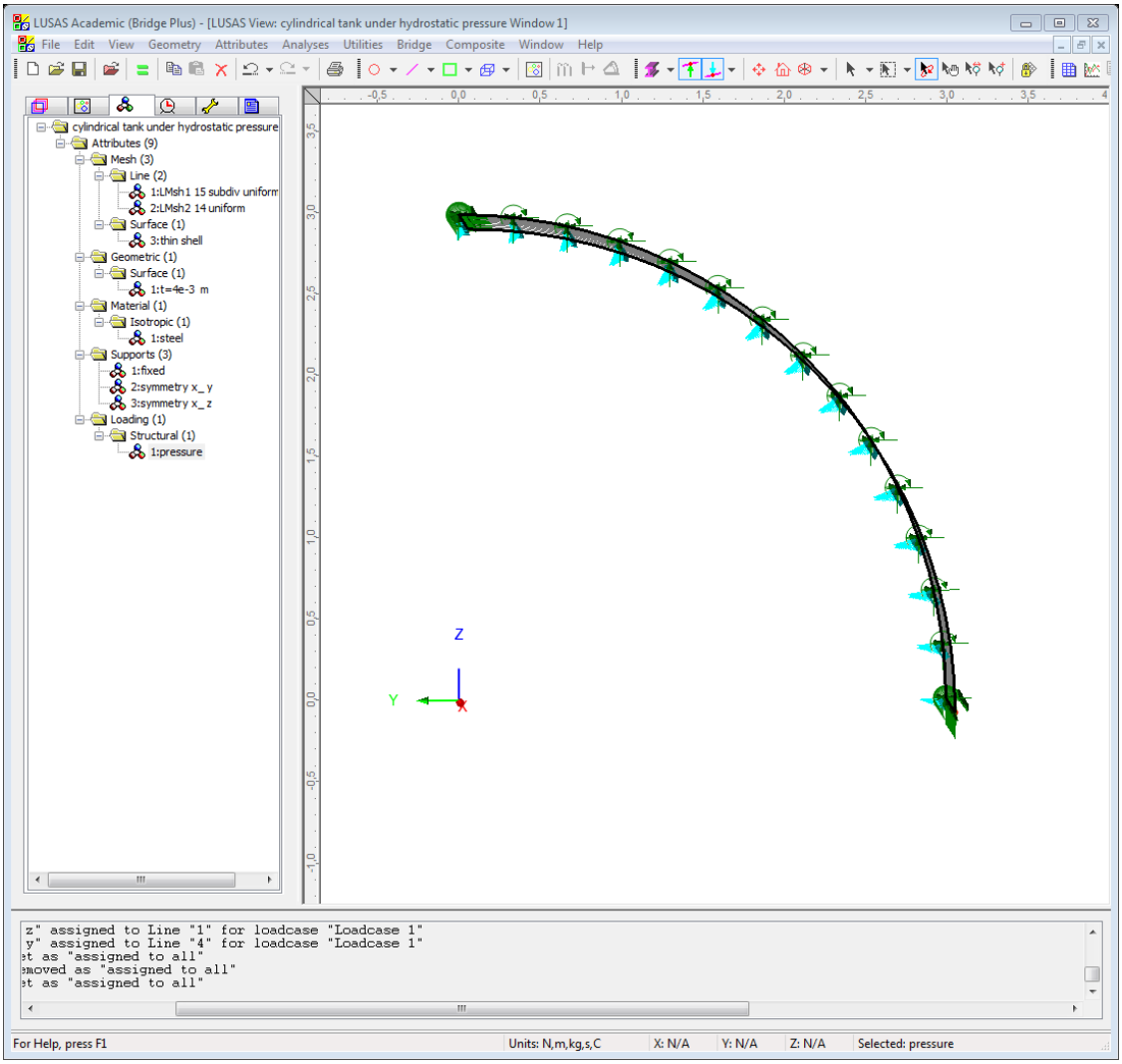
- Assign the loads: !!!pay attention to which line:





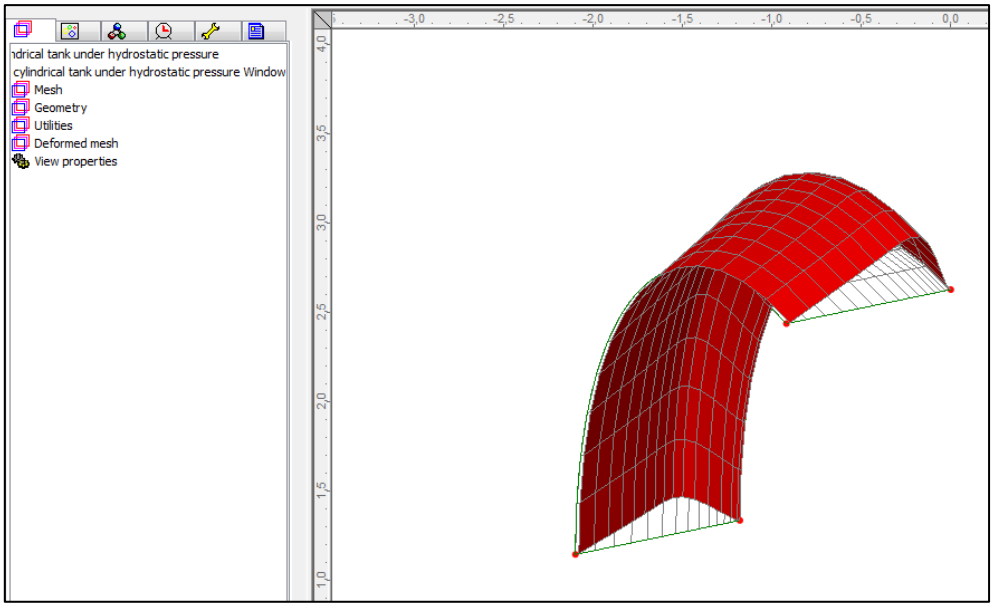


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 65 di 104

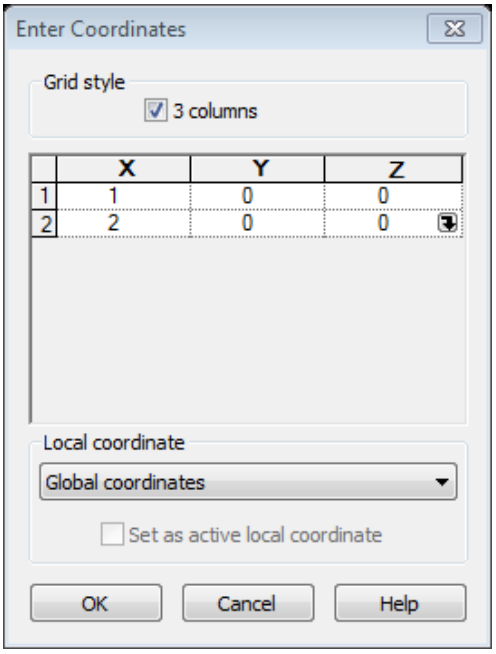


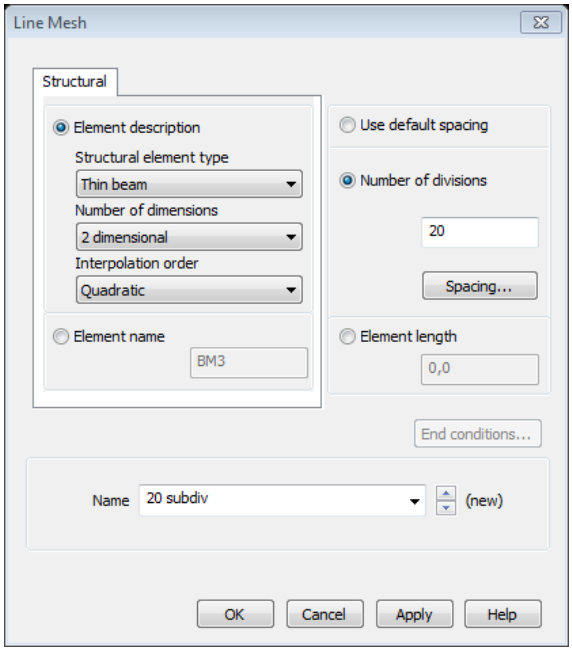
RUN!!!

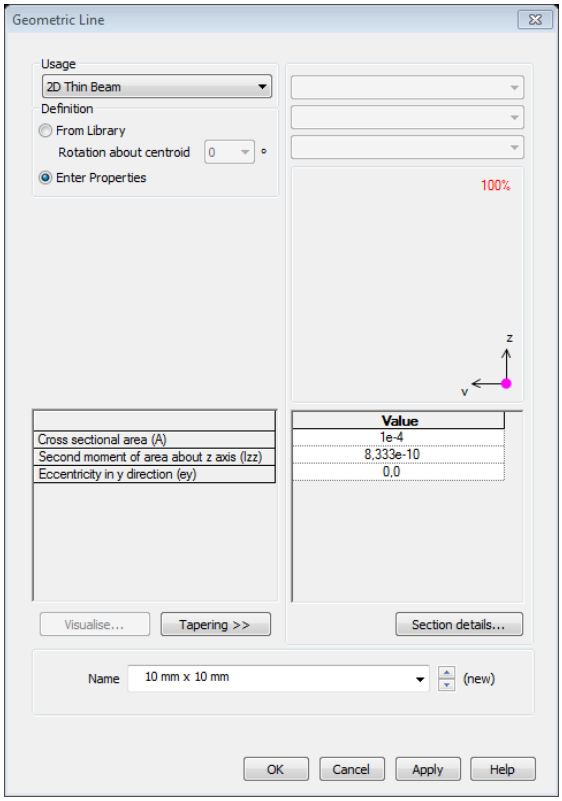
Deformed mesh:

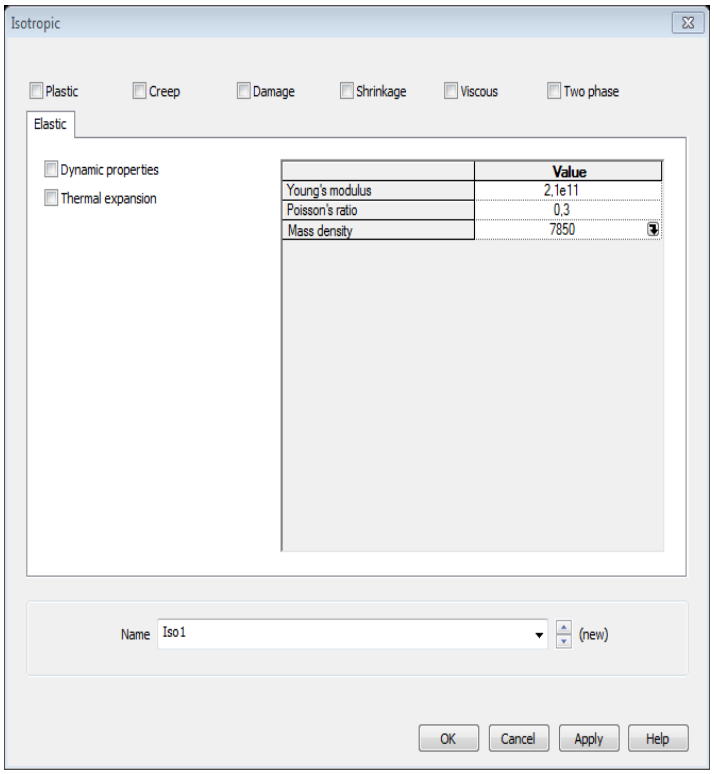


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 67 di 104







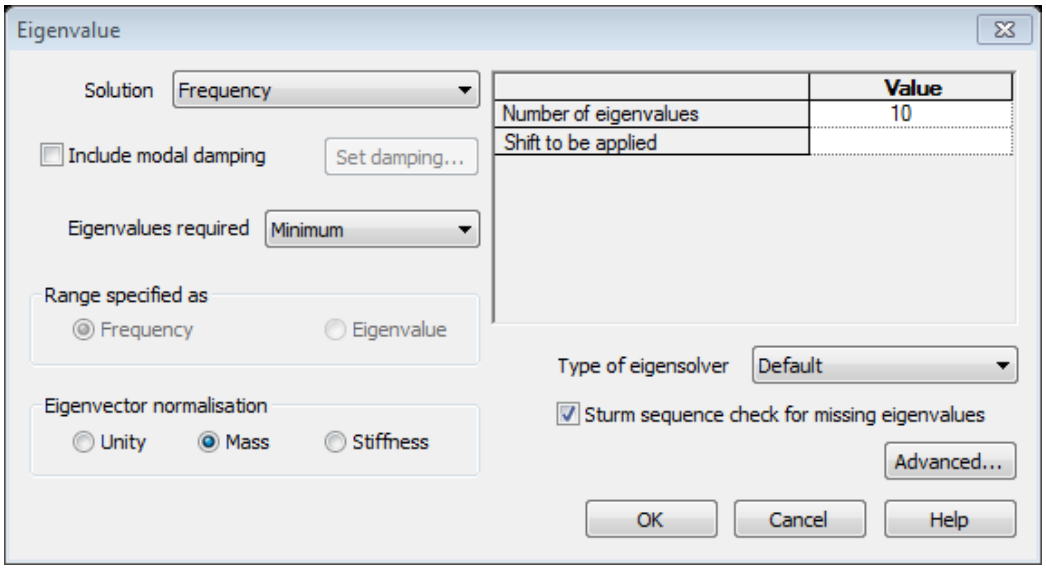


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 69 di 104

!!! N= nr of elements (coming from the beam subdivisions). We can ask up to (N-1) eigenvalues .

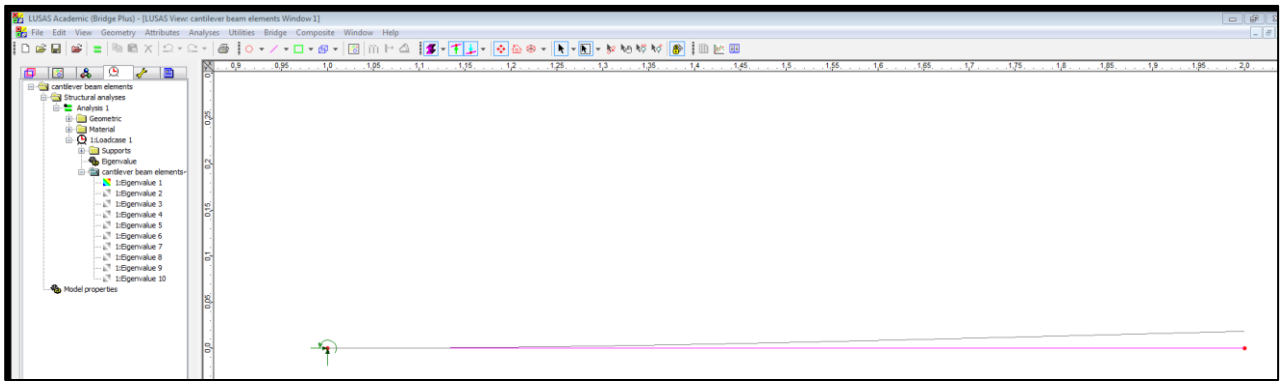
We divided into 20 but we ask 10 eigenvalues (only the first ones will be well described very well)

"Shift" means that the start frequency is shifted: ex. if "shift to be applied=30 Hz", then it starts from 30 Hz.

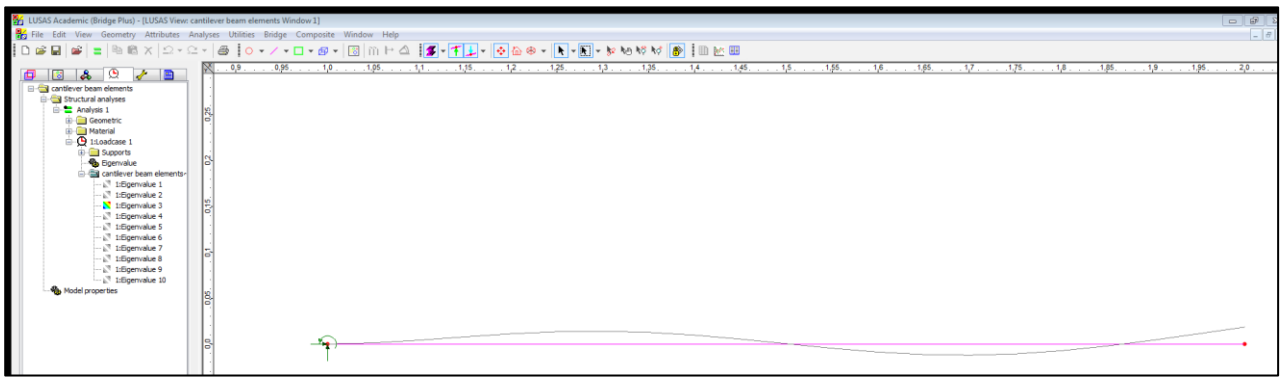


RUN!!!

- **Eigenvalue 1:**



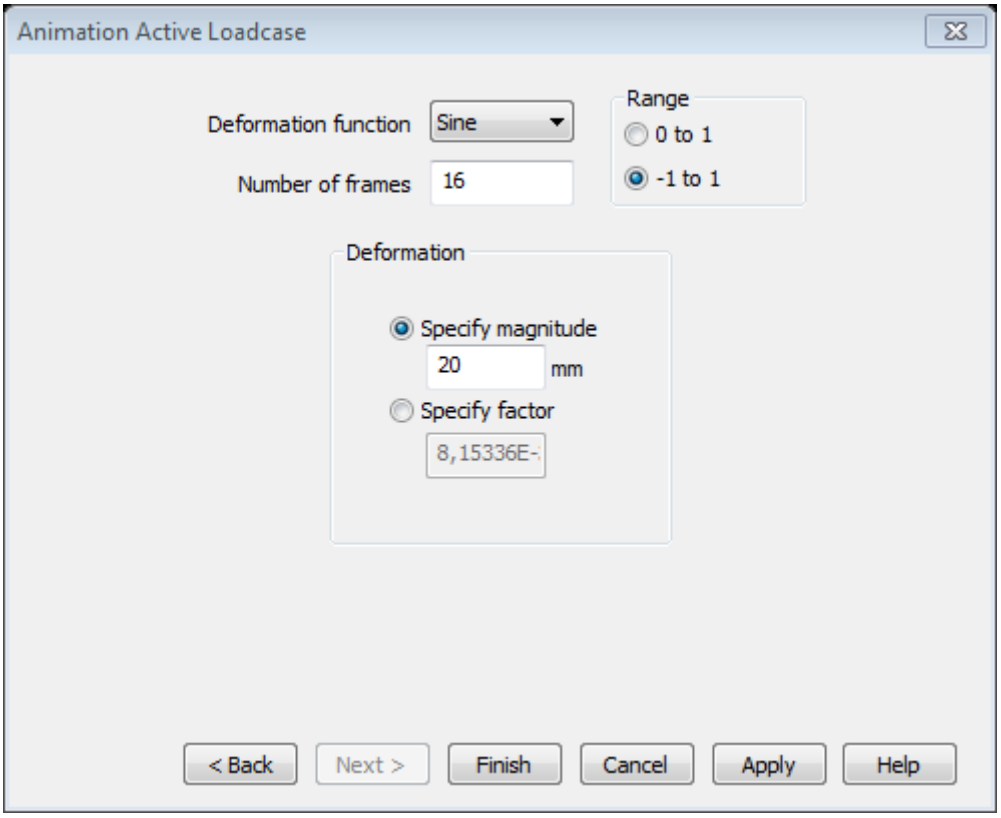
- **Eigenvalue 3:**

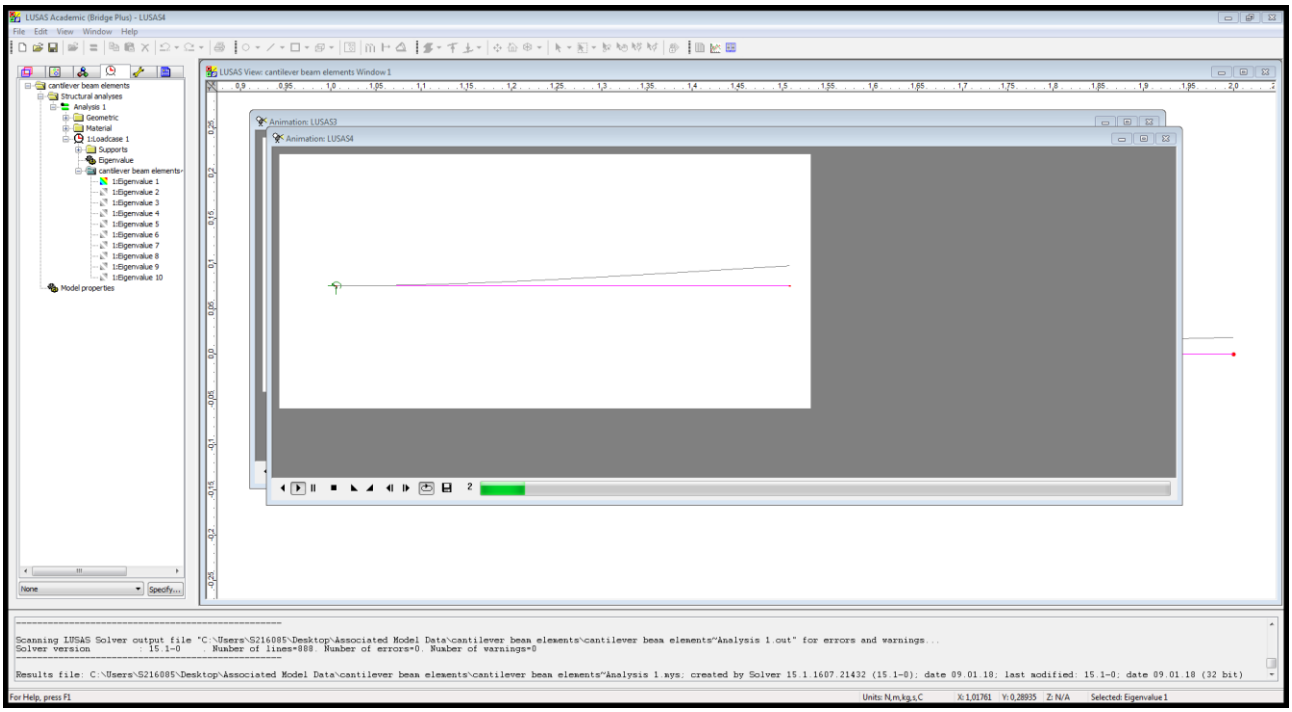


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 71 di 104

- **How to see the video of the movement:**

- Utility, animation wizard, next, nr of frames 16, from -1 to 1 (complete cycle)



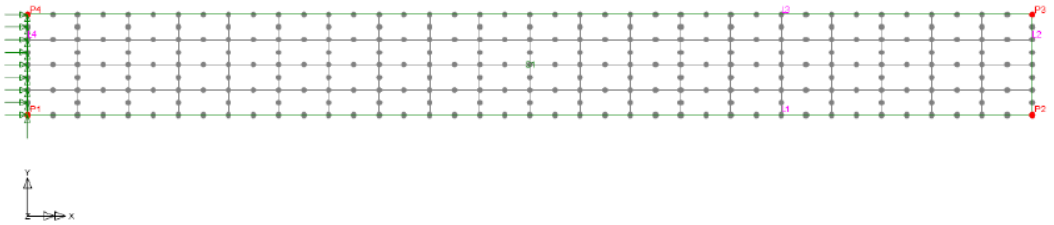


© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 73 di 104

Dynamics: Plane stress elements 2D

Same beam as in the previous example with cross section 10 cm x 10 cm.

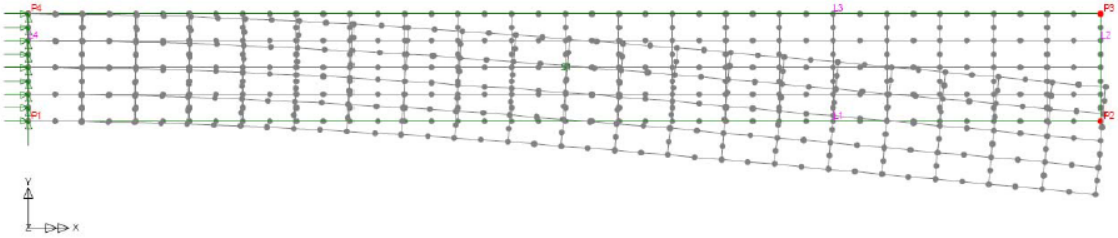
For the solution, use quadrilateral *plane stress* elements, with *quadratic interpolation* and a subdivision in 20 x 4 elements.



NUMERICAL RESULTS:

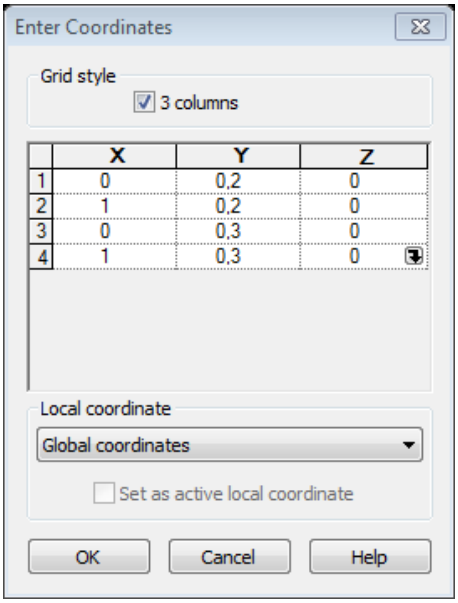
- Mode shapes (and corresponding stress contours):

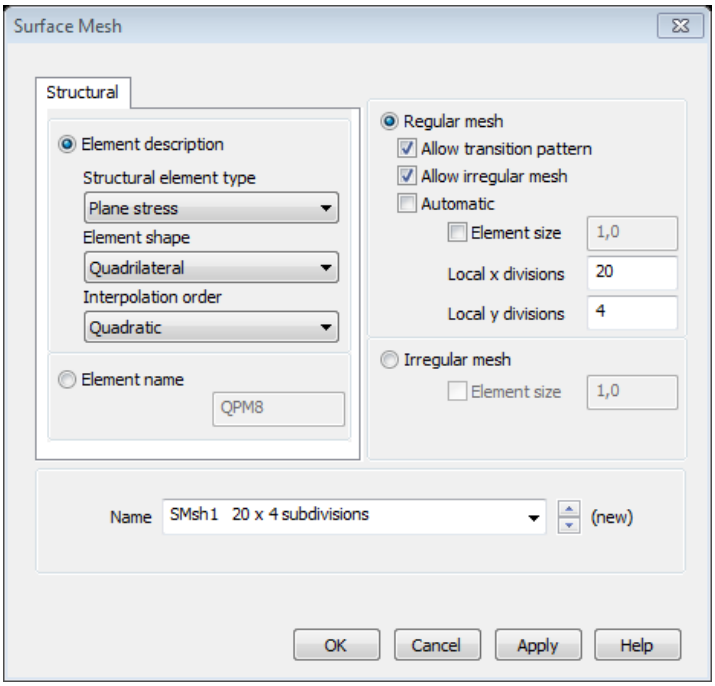
Mode 1

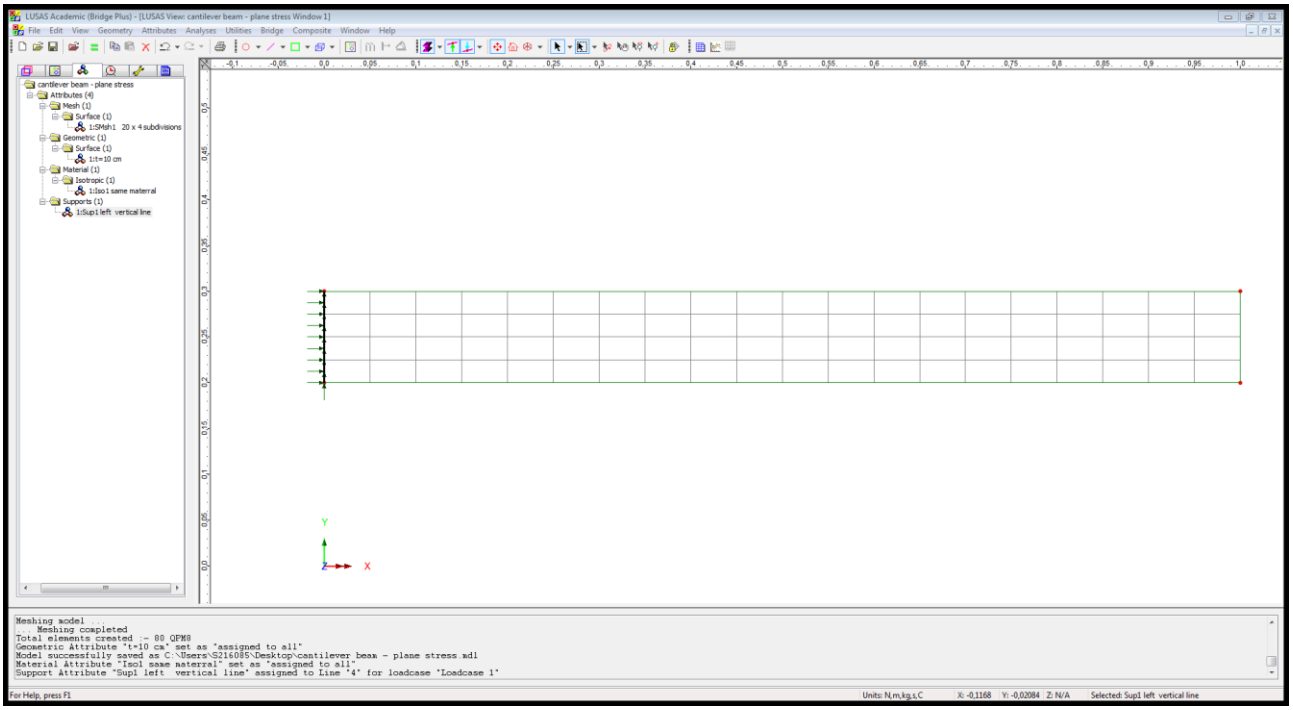


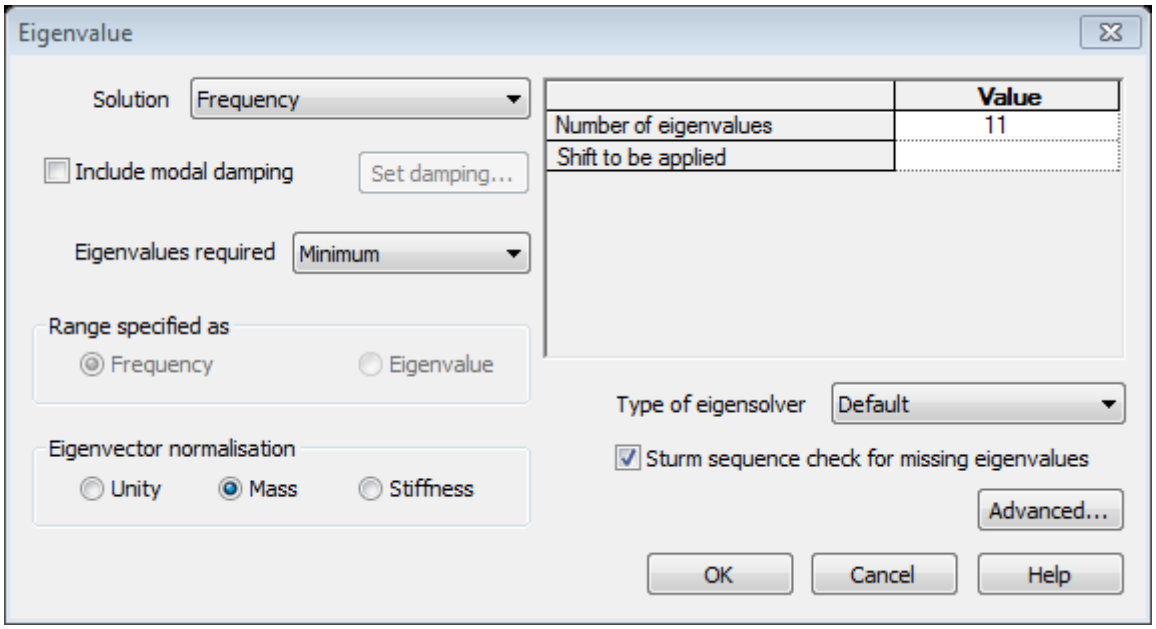
- **Solution:**

-New points and surface





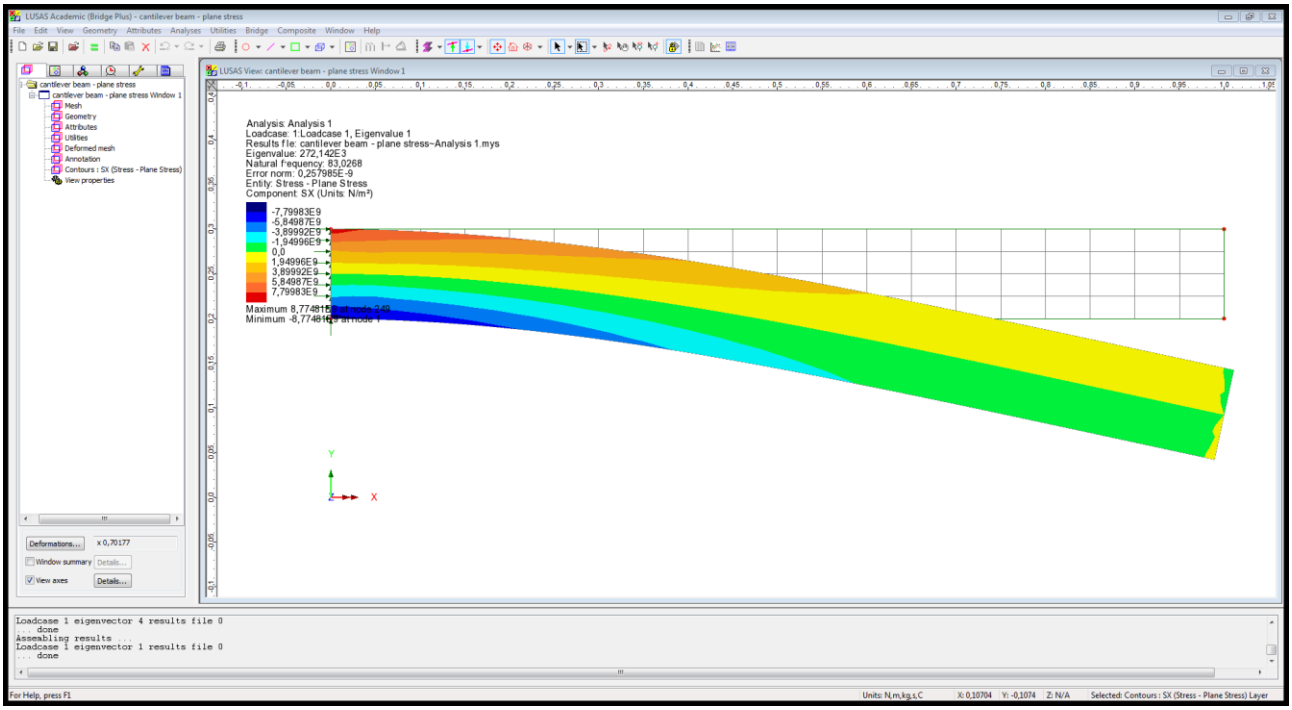




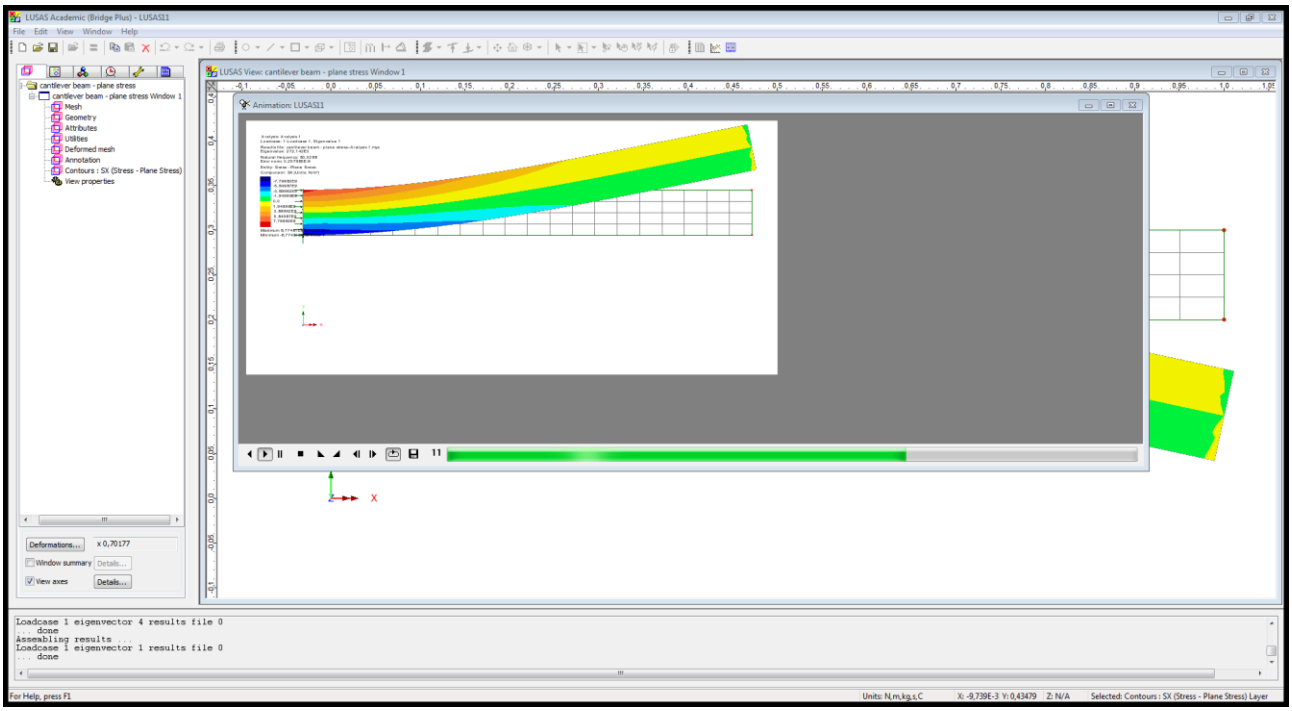
RUN!!!

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 77 di 104

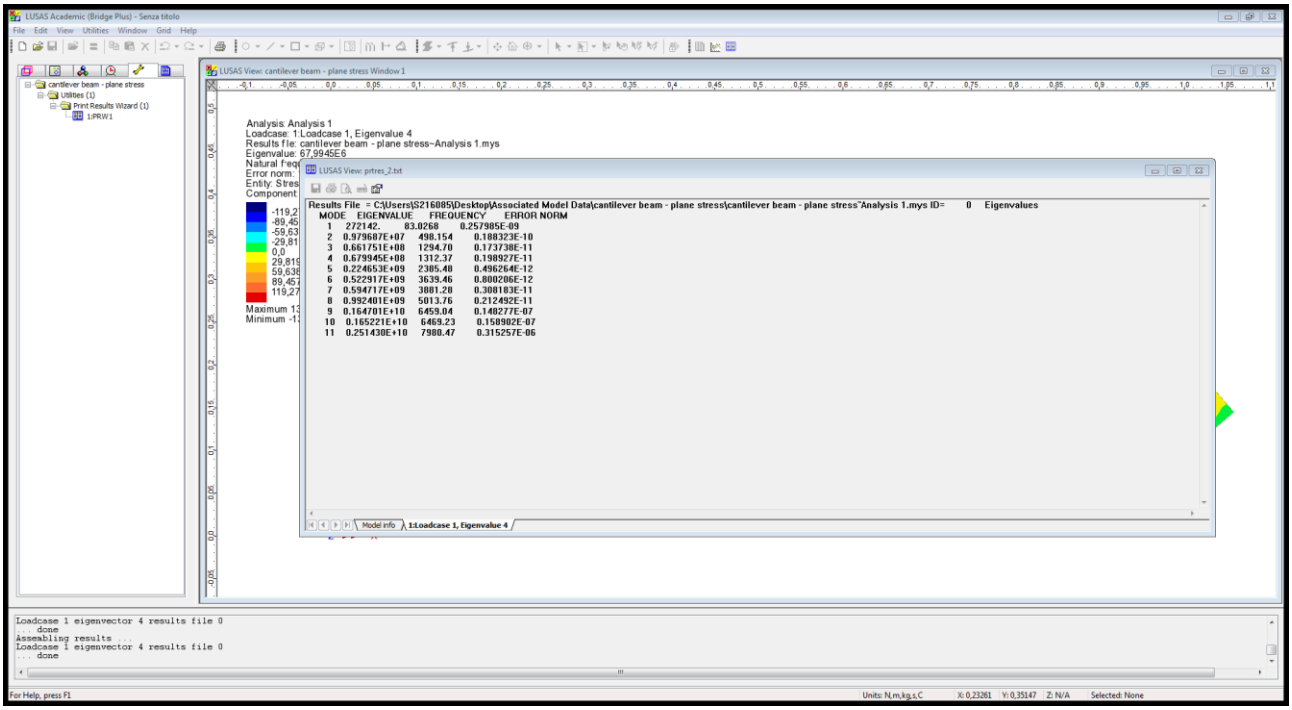
Eigen 1



Utilities, animation wizard,

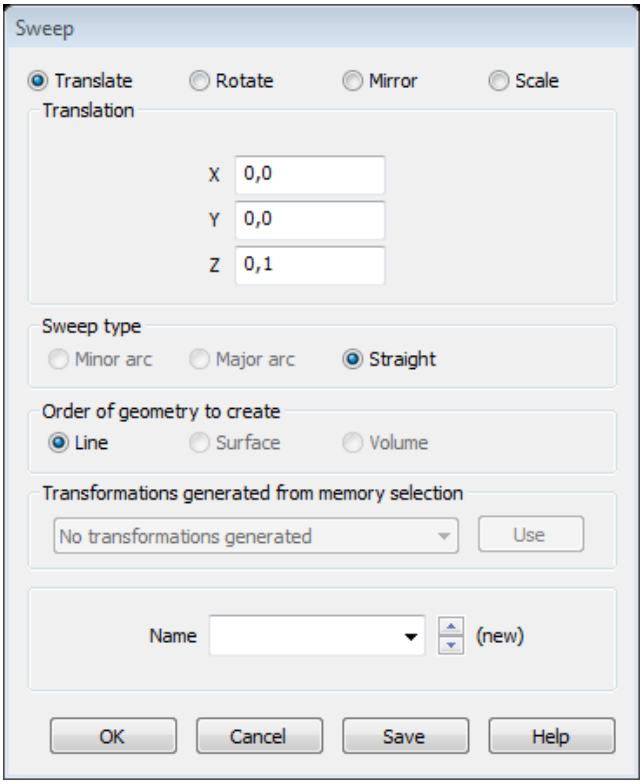


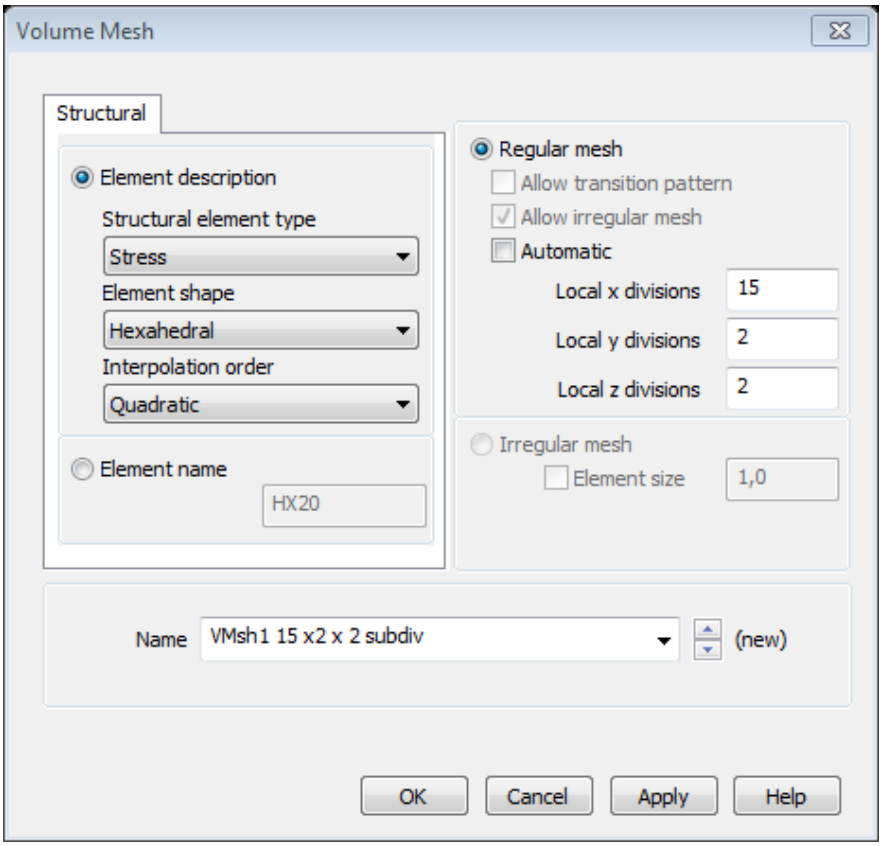
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 79 di 104

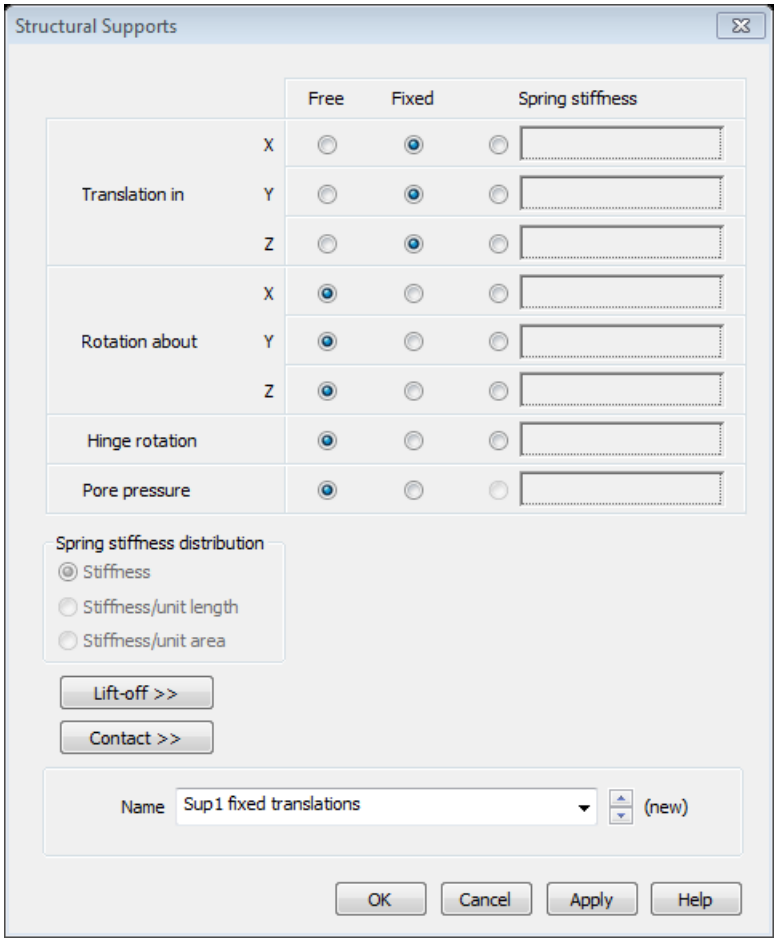


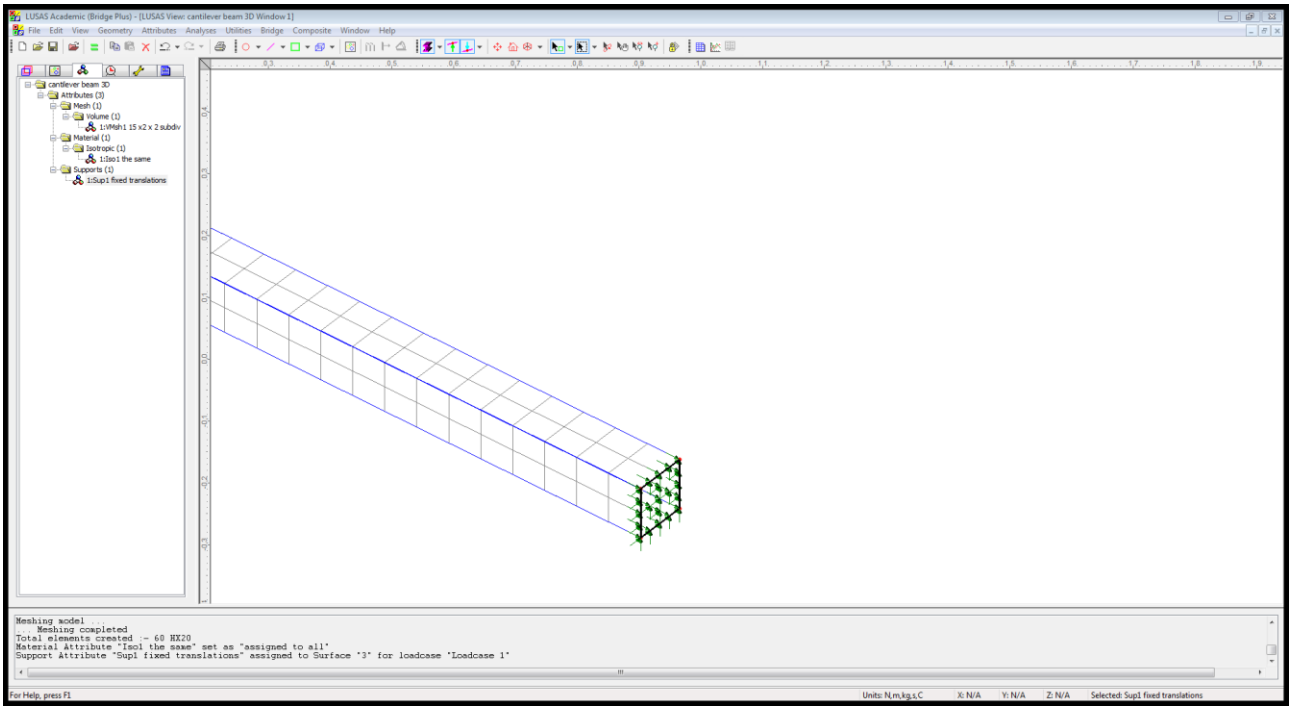
© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 81 di 104

- **How to draw the solid:**
  - Start from the previous cantilever beam and sweep it:



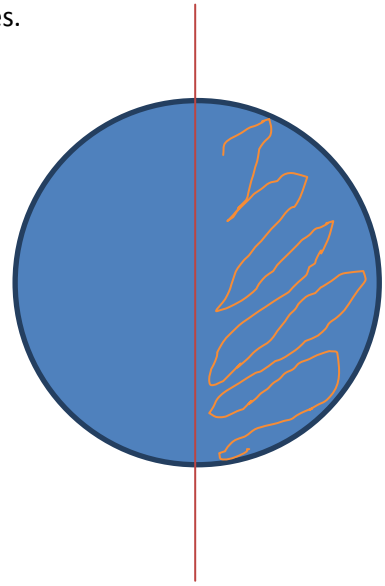




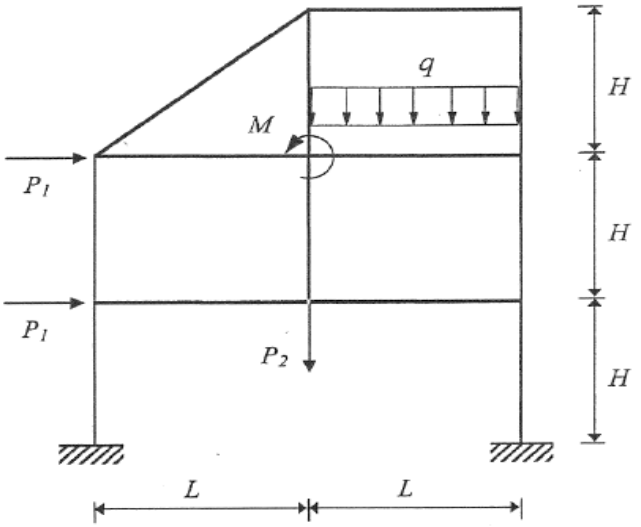


Dynamics: Free vibrations of frames

!!! For vibration (in dynamics) we need at least half plate (not only one quarter as we did before) so we don't lose the anti-symmetric modes.



(Same structure proposed for running a static analysis in LAIB 2017_10_26)



$L = 5$ m $\quad H = 3$ m

$P_1 = 10$ kN $\quad P_2 = 20$ kN $\quad M = 10$ kNm $\quad q = 10$ kN/m

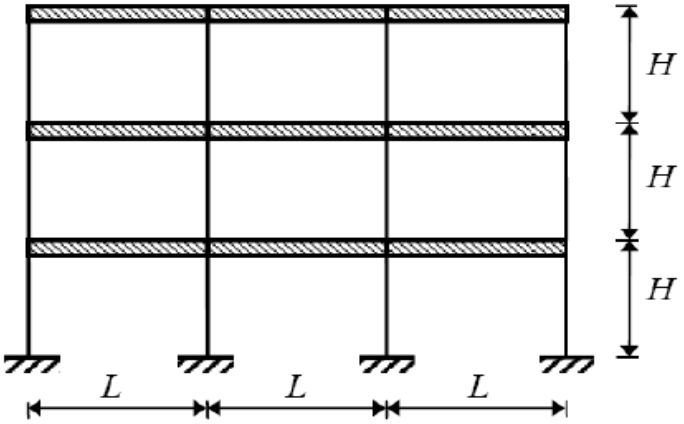
Concrete: $E = 2.9\ 10^{10}$ N/m^2 $\quad \nu = 0.2$ $\quad \rho = 2500$ kg/m^3

First floor columns: bx$h = 50$x50 cm^2 ($A = 0.25$ m^2; $I_{zz} = 5.20833\ 10^{-3}$ m^4)

Second floor columns: bx$h = 40$x40 cm^2 ($A = 0.16$ m^2; $I_{zz} = 2.1333\ 10^{-3}$ m^4)

Third floor columns: bx$h = 30$x30 cm^2 ($A = 0.09$ m^2; $I_{zz} = 0.6750\ 10^{-3}$ m^4)

Dynamics: Plane frame shear type



Columns and beams have rectangular cross-section (width b, depth h).

$L = 5$ m $\quad H = 3$ m

$E = 2.9\ 10^{10}$ N/m^2 $\quad \nu = 0.2$ $\quad \rho = 2500$ kg/m^3

First floor columns: $bxh = 50x50$ cm^2 ($A = 0.25$ m^2; $I_{zz} = 5.20833\ 10^{-3}$ m^4)

Second floor columns: $bxh = 40x40$ cm^2 ($A = 0.16$ m^2; $I_{zz} = 2.1333\ 10^{-3}$ m^4)

Third floor columns: $bxh = 30x30$ cm^2 ($A = 0.09$ m^2; $I_{zz} = 0.6750\ 10^{-3}$ m^4)

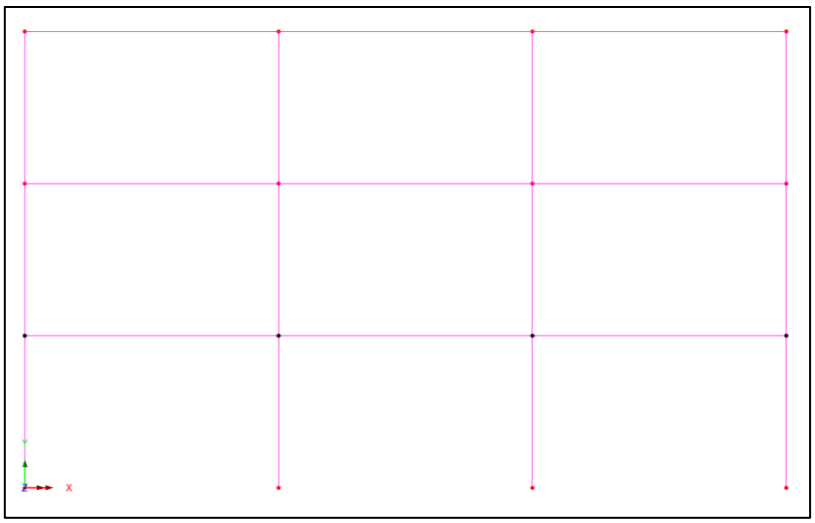
Beams: $bxh = 30x70$ cm^2 ($A = 0.21$ m^2; $I_{zz} = 8.5750\ 10^{-3}$ m^4)

Beams are much stiffer than the columns: EI(beam) **>** EI(column).

In an ideal shear type frame, the EI(beam) goes to infinity.

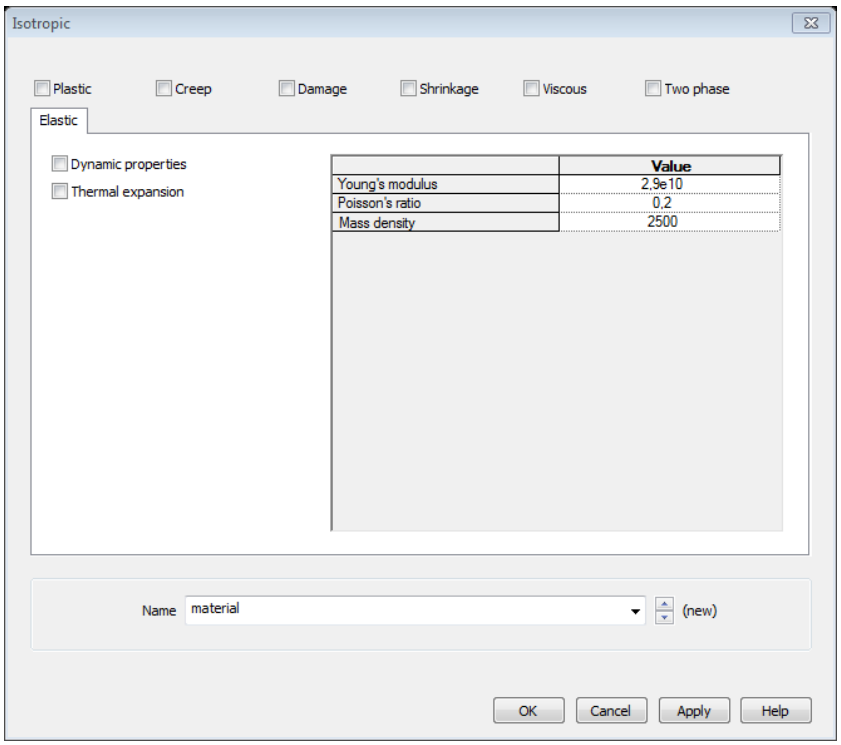
- **How to draw the frame:**

- Insert the points and connect by lines:

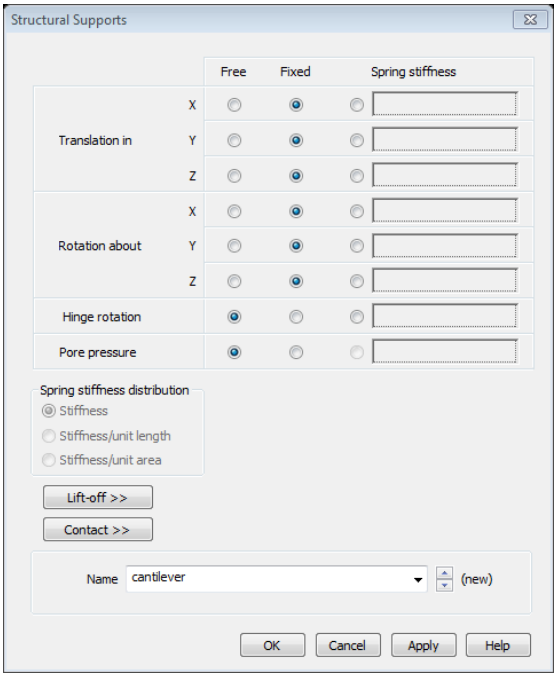


- Assign mesh : 10 subdivisions to each line

-Insert material:



- Assign cantilever support:



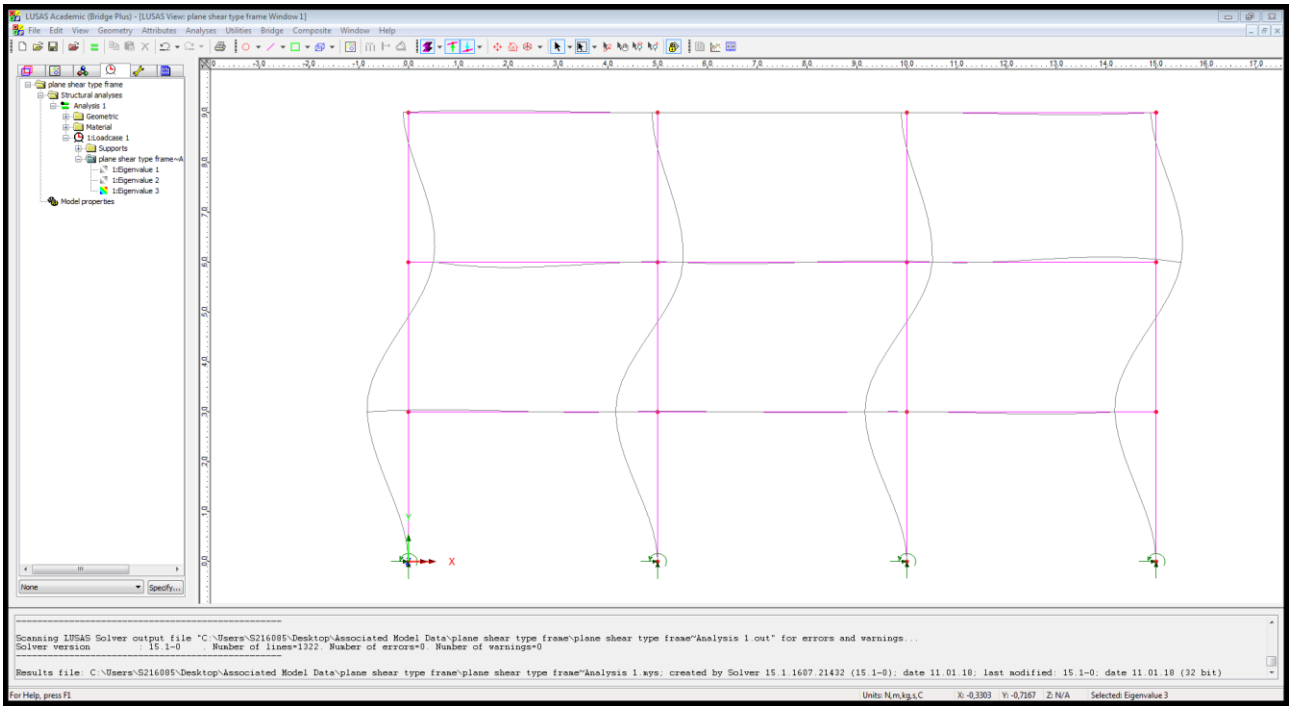
- No load is applied, but only the loadcase1 with 3 eigenvalues: **N = 3 eigenvalues = nr of floors**.

[if nr of floors is 2 then N=2]

RUN!!!

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 91 di 104

- **Mode 3:**



- See the values: *utilities, print result wizard, next, none, ok*

N	Eigenvalue	Natural frequency	Error
1	1597.25	6.36073	0.129864E-10
2	8719.54	14.8616	0.447221E-11
3	33523.0	29.1401	0.143019E-05

© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 93 di 104

Displacement unit of measure? [m]

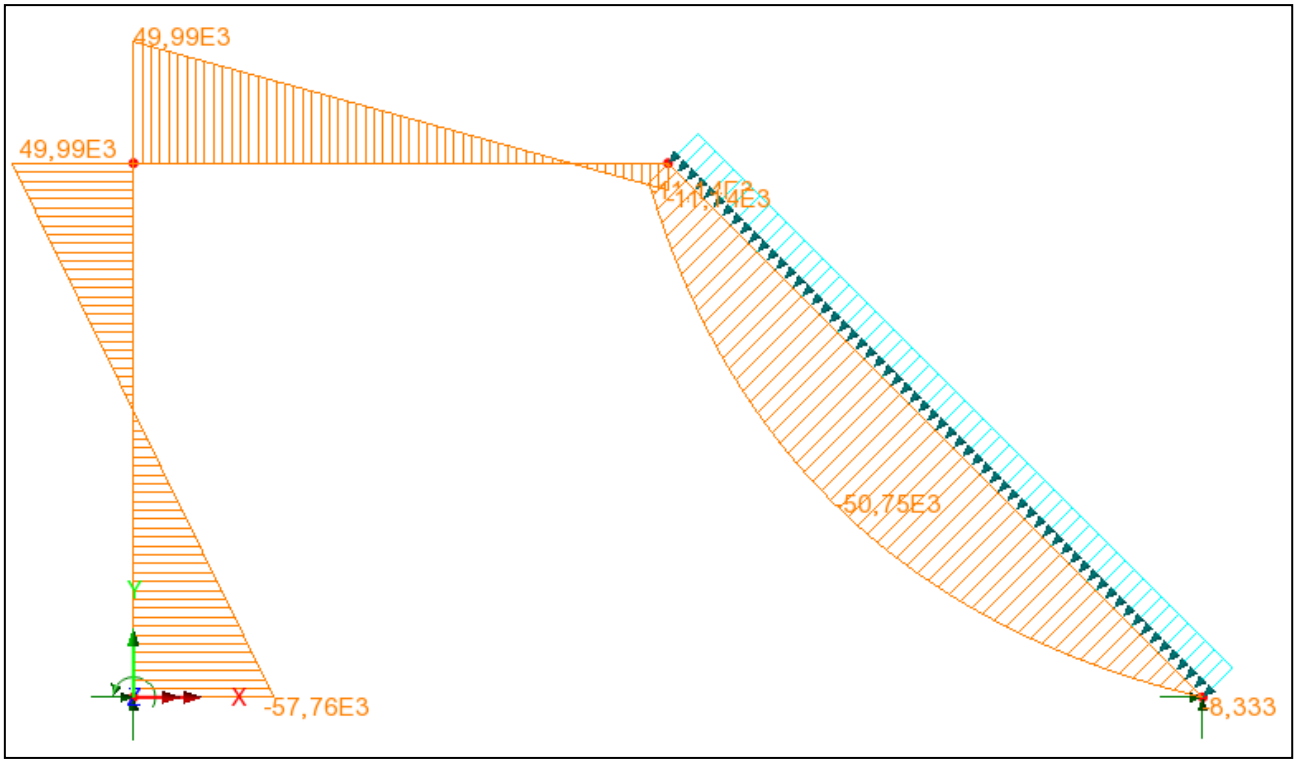
THIN BEAM:

Max displacement (resultant) = 88,0899

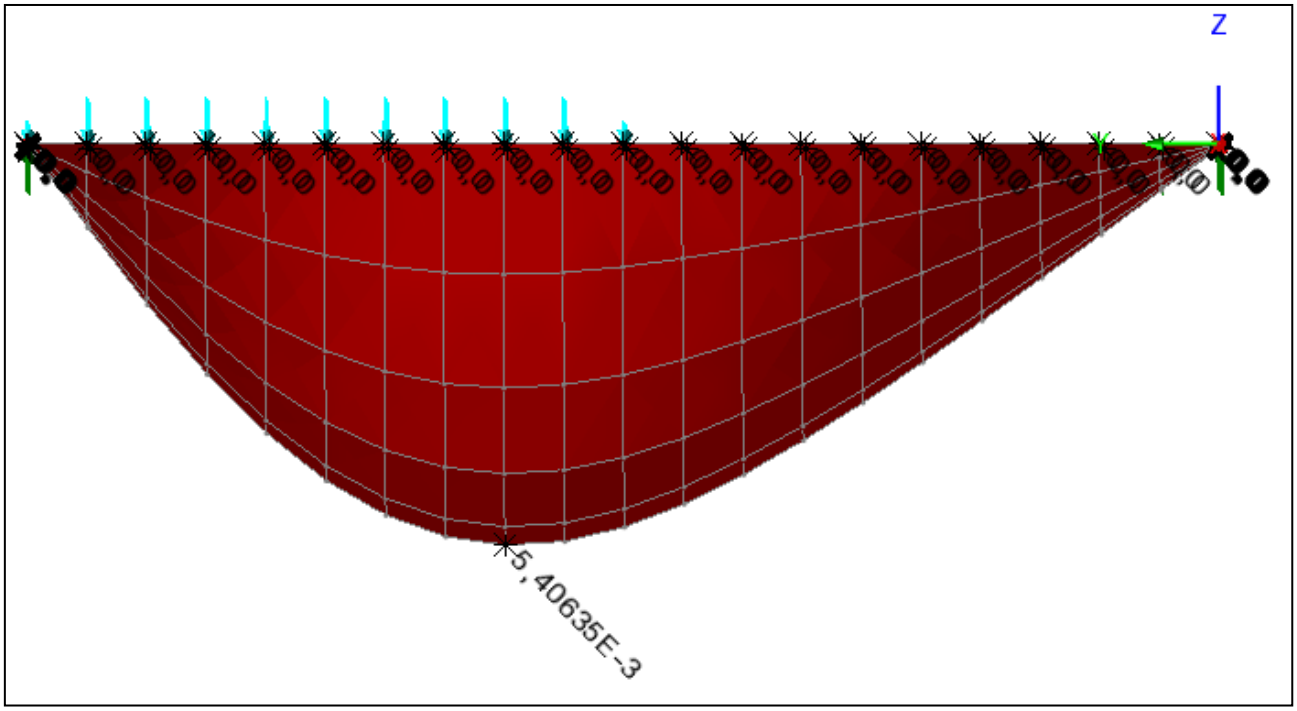
Max displ (Dx) = -62,2881

Max displ (Dy) = -62,2898

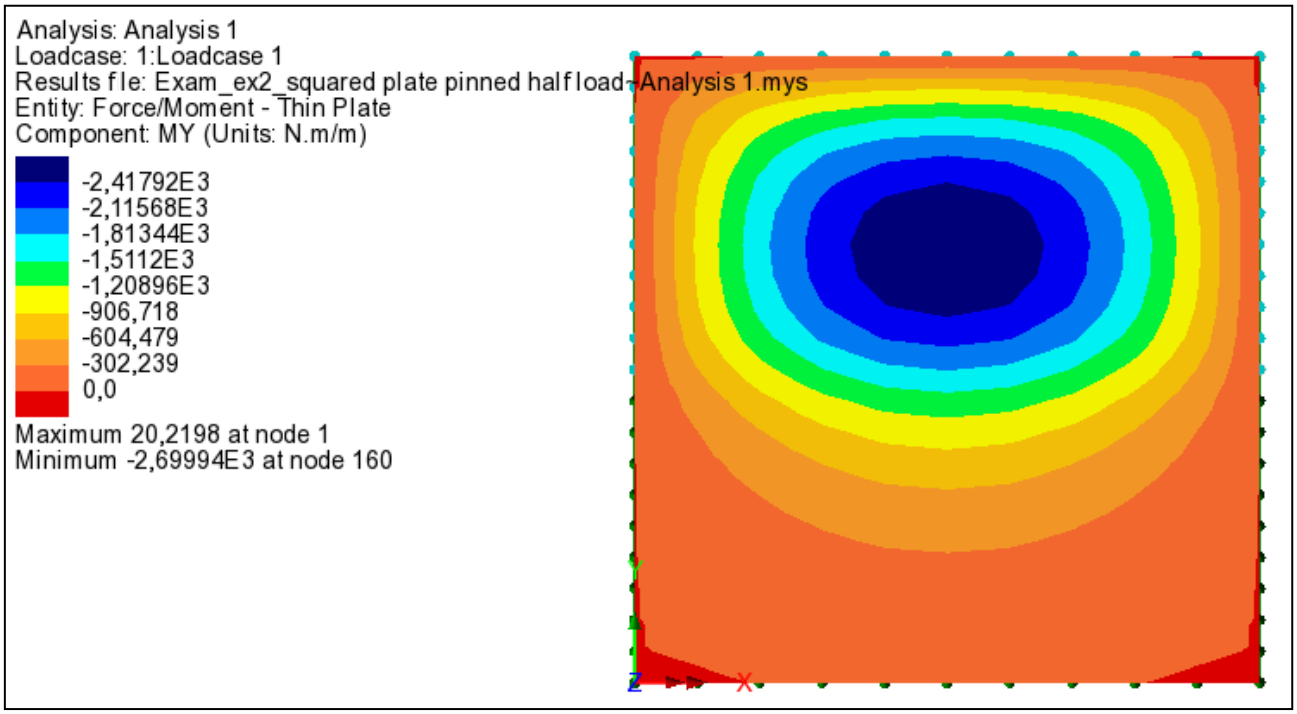
Maximum moment (Mz) in abs value = 57,76 * 10^3 in the fixed extreme of the vertical beam:



- Deformed mesh:



- Diagram of moments My:



© Proprietà riservata dell'autore - Digitalizzazione e distribuzione a cura del CENTRO APPUNTI - Corso Luigi Einaudi, 55 - Torino / Pagina 97 di 104