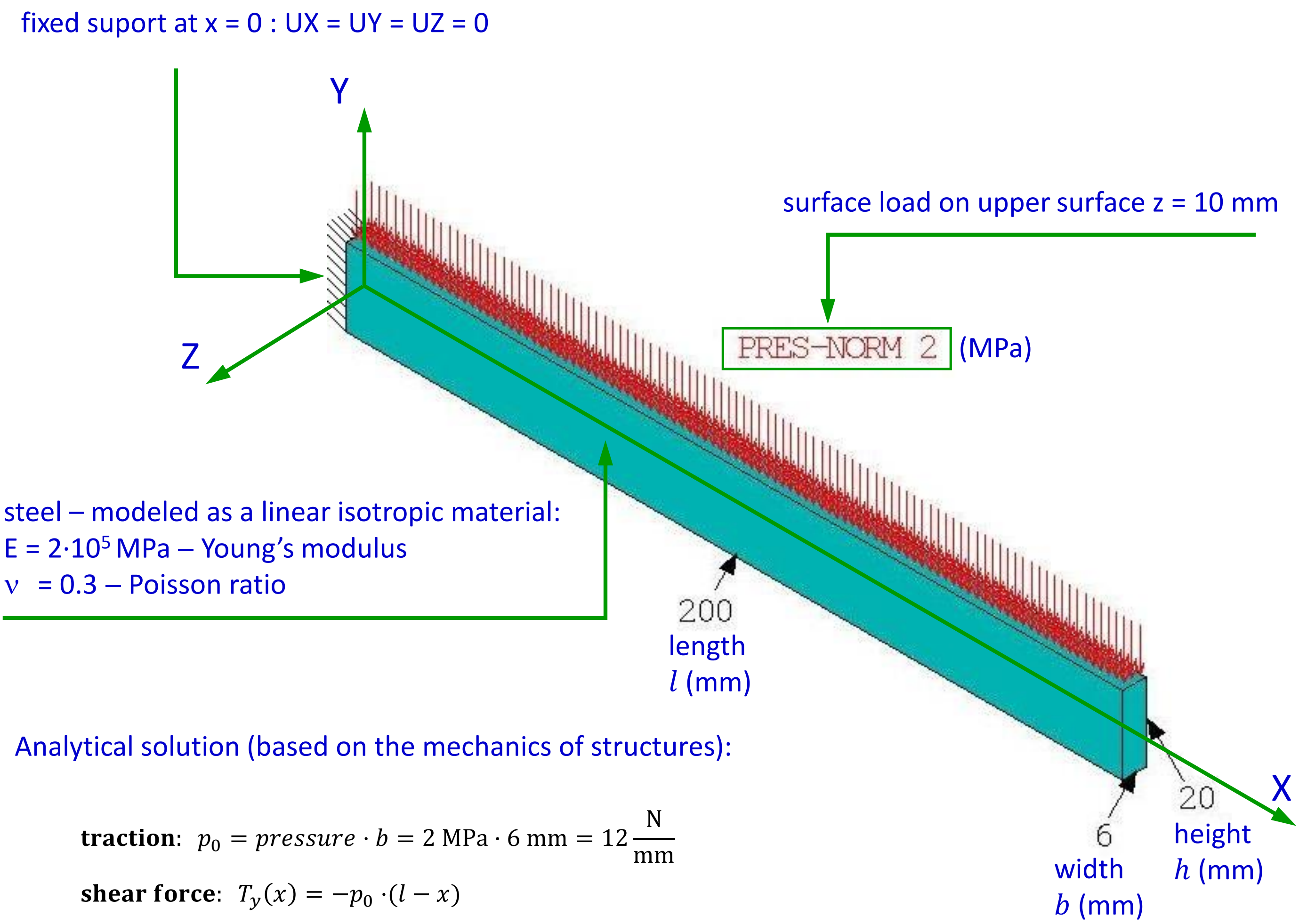
2D FE model of a centilever beam loaded by pressure



Analytical solution (based on the mechanics of structures):

traction: $p_0 = pressure \cdot b = 2\text{ MPa} \cdot 6\text{ mm} = 12\,\frac{\text{N}}{\text{mm}}$

shear force: $T_y(x) = -p_0 \cdot (l - x)$

$T_y(0) = -12\,\frac{\text{N}}{\text{mm}} \cdot (200\text{mm}) = -2400\text{ N}$; $T_y(\frac{l}{2}) = -1200\text{ N}$

bending moment: $M_z(x) = -p_0 \frac{(l-x)^2}{2}$

$M_z(0) = -12\,\frac{\text{N}}{\text{mm}} \cdot \frac{(200mm)^2}{2} = -2.4 \cdot 10^5\text{ Nmm}$; $M_z\,(\frac{l}{2}) = -6 \cdot 10^4\text{ Nmm}$; $M_z\,(l) = 0\text{ Nmm}$

second moment of area: $J_z = b\,\frac{h^3}{12} = 6\text{mm} \cdot \frac{(20\text{mm})^3}{12} = 4000\text{ mm}^4$

normal stress: $\sigma_x(x,y) = -M_z(x) \cdot \frac{y}{J_z}$

$\sigma_x(0, \frac{h}{2}) = -2.4 \cdot 10^5\text{ Nmm} \cdot \frac{10\text{ mm}}{4000\text{ mm}^4} = 600\text{ MPa}$; $\sigma_x(\frac{l}{2}, \frac{h}{2}) = -6 \cdot 10^4\text{ Nmm} \cdot \frac{10\text{ mm}}{4000\text{ mm}^4} = 150\text{ MPa}$

shear stress: $\tau_{xy}(x,y) = \frac{3T_y(x)}{2bh}(1 - \frac{4z^2}{h^2})$

$\tau_{xy}\left(\frac{l}{2}, 0\right) = \frac{3 \cdot (-1200\text{ N})}{2 \cdot 6\,mm \cdot 20\,mm} = -15\text{ MPa}$; $\tau_{xy}\left(\frac{l}{2}, \pm\frac{h}{2}\right) = \frac{3 \cdot (-1200\text{ N})}{2 \cdot 6\,mm \cdot 20\,mm}(1-1) = 0\text{ MPa}$
$= -30\text{ MPa}$

deflection at x = $\boldsymbol{l}$: $v(l) = -\frac{p_0 \cdot l^4}{8 \cdot E \cdot J_z} = -\frac{12\frac{\text{N}}{\text{mm}} \cdot (200\text{mm})^4}{8 \cdot 2 \cdot 10^5\text{MPa} \cdot 4000\text{mm}^4} = -3\text{ mm}$

Comment:

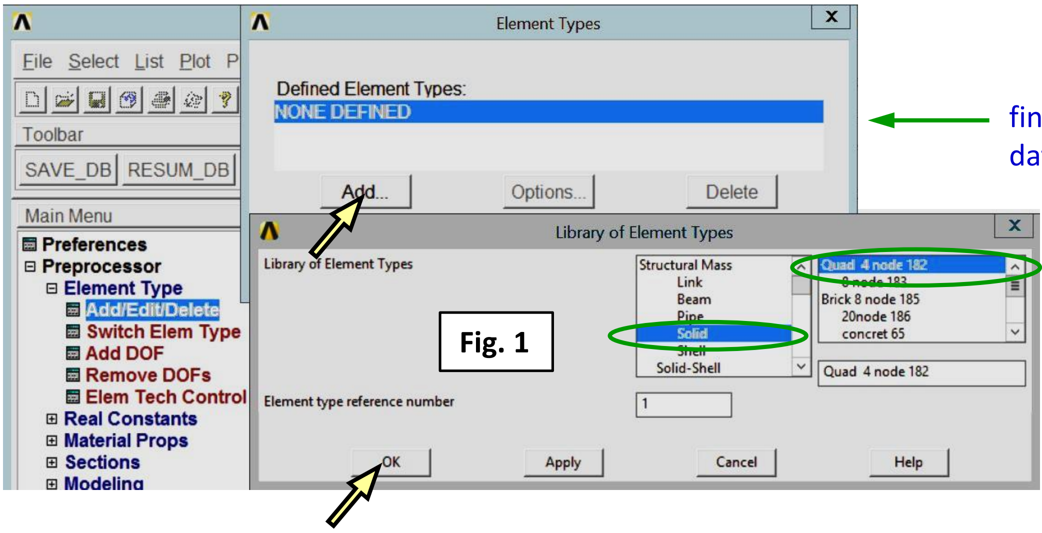
The problem can be solved as a 2D structural structure, assuming a plane stress condition. Numerical results will be compared with the analytical solution.

Clear and start a new database

Utility Menu > File > Clear & Start New > Do not Read File > OK > CLEAR ... EXECUTED? > Yes
Utility Menu > Plot > Replot

Choose the element type

Main Menu > Preprocessor > Element Type > Add → OK > Solid > Quad 4 node 182 → OK (Fig. 1)

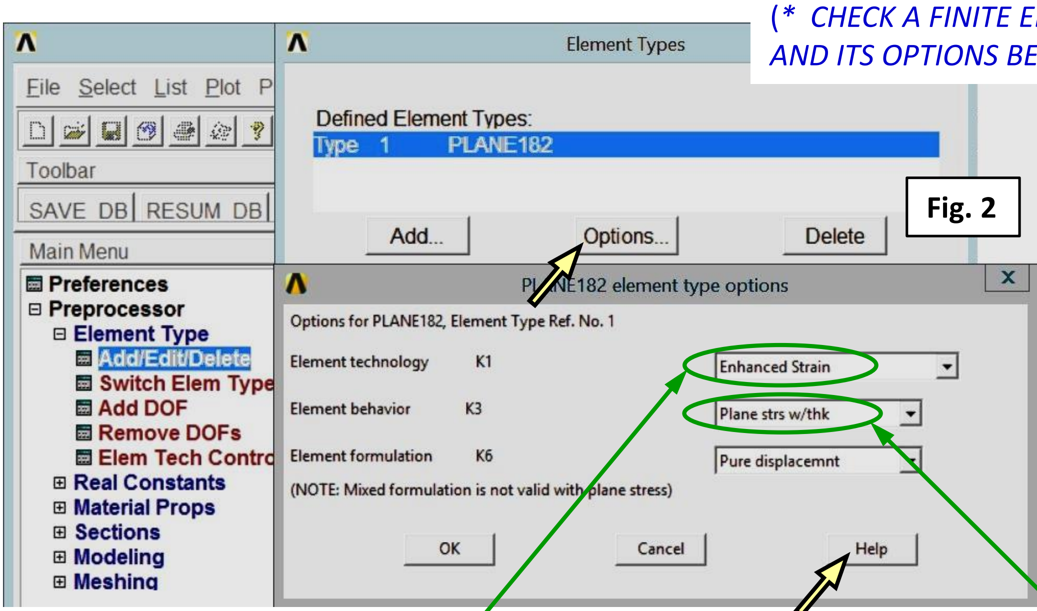


finite elements used in the database

library of elements available in the ANSYS system

Main Menu > Preprocessor > Element Type > Options → OK > Element Technology K1 = Enhanced Strain > Element Behavior K3 = Plane stress w/thk → Help (Fig. 2) *

(CHECK A FINITE ELEMENT AND ITS OPTIONS BEFORE USE)*



Information from ANSYS Help Viewer

… **PLANE182** is used for **2-D modeling of solid structures**. The element can be used as either **a plane element (plane stress,** plane strain or generalized plane strain) or an axisymmetric element. It is defined by **four nodes** having **two degrees of freedom at each node: translations in the nodal x and y directions**. … PLANE182 Element Technology … For more information, see **Element Technologies** > 5.1.2. Element Technologies > Current-Technology > 2.4.1. Legacy vs. Current Element Technologies > Automatic Selection of Element Technologies and Formulations> Table 5.4: Recommendation Criteria for Element Technology (**Linear Material**) > Plane stress **> KEYOPT(1) = 2** (Enhanced Strain).

assumed linear isotropic properties of a steel

Close ANSYS Help Viewer and „PLANE 182 elem. type options”→ OK, and „Element Type” → Close

Define the beam thickness as a real constant

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete > Add...> Type 1 PLANE 182 → OK (Fig. 3)
Real Constant Set. No. = 1, Thickness THK = 6 → OK > Close (Fig. 4)

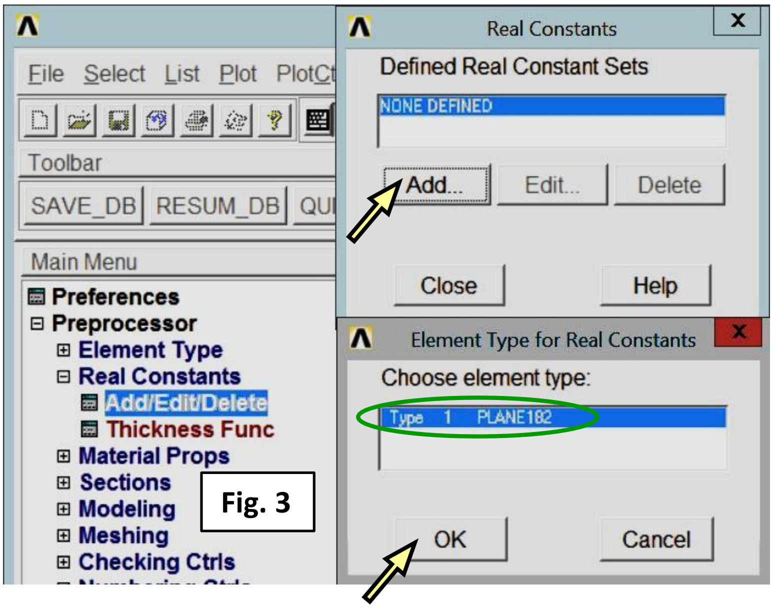


Fig. 3

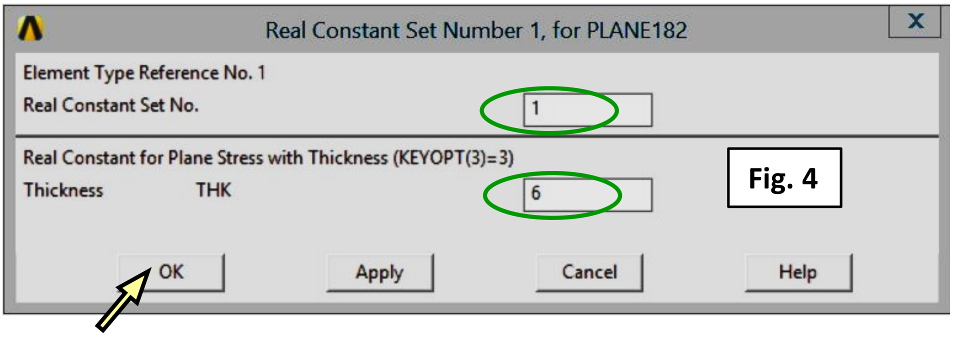
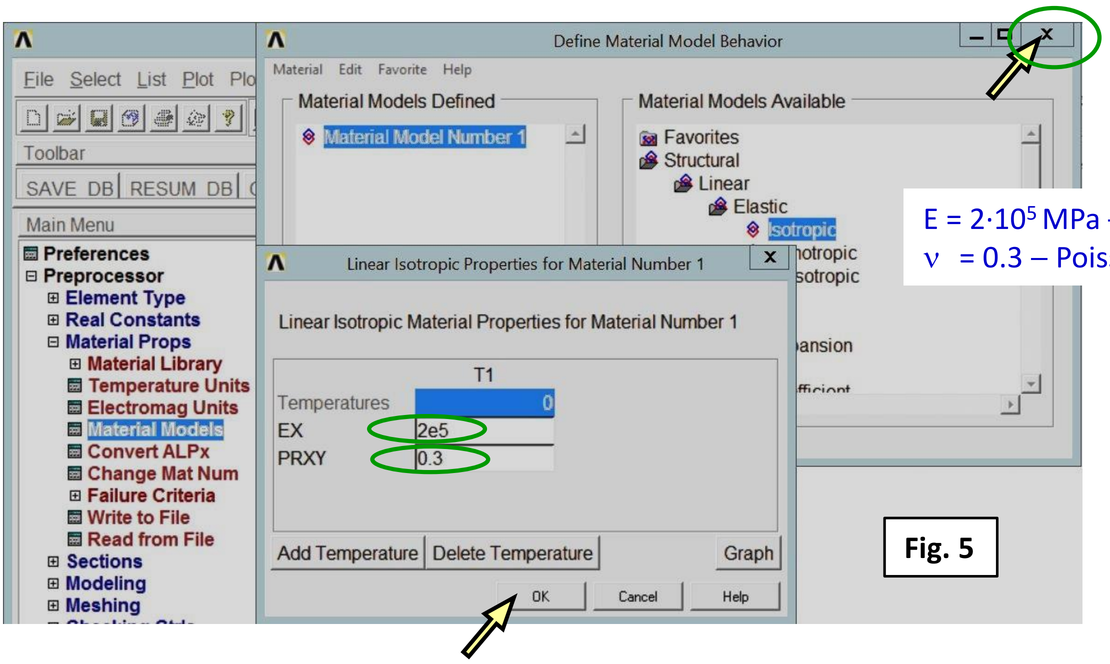


Fig. 4

Define Material Properties

Main Menu > Preprocessor > Material Props > Material Models > Material Model Number 1 > Structural > Linear > Elastic > Isotropic > EX = 2e5, PRXY = 0.3 → OK > Close (Fig. 5)



E = 2·10^5 MPa – Young's modulus
ν = 0.3 – Poisson ratio

Fig. 5

Create a rectangle

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions
X1, X2 → 0, 200 ; Y1, Y2 → −10, 10 → OK (Fig. 6)

Save a database file

Utility Menu > File > Save as…

beam_model.db

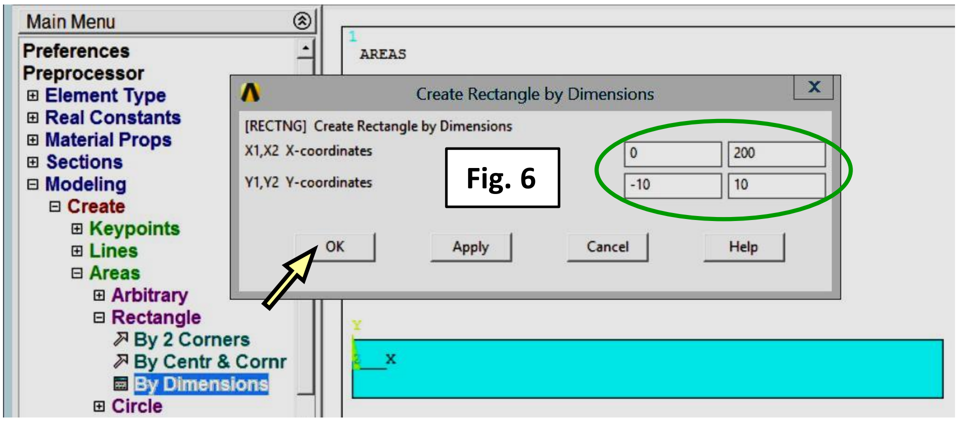


Fig. 6

<u>Define a discrete model</u>

Main Menu > Preprocessor >
> Meshing > Mesh Tool (Fig. 7)

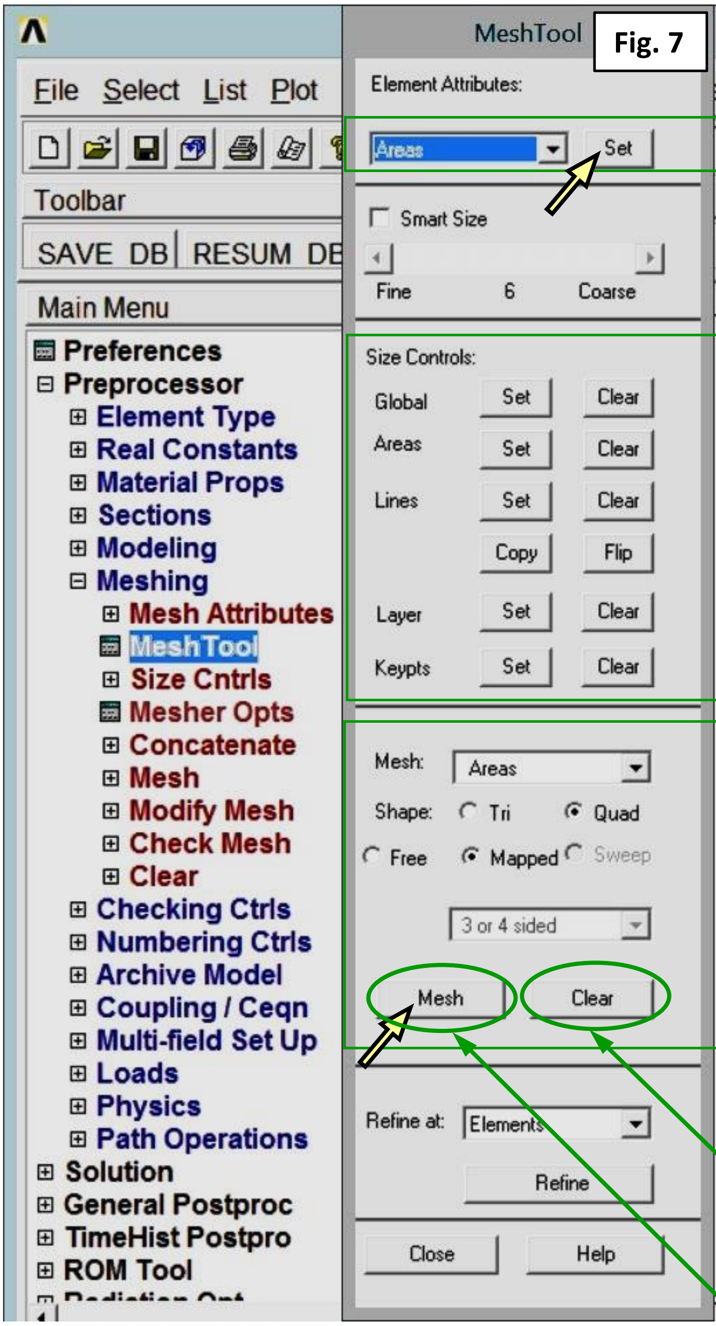


Fig. 7

ATTRIBUTES

- material
- real constant
- element type
- section
- element coordinate system

<u>Comment: Attributes can be assigned or changed only if the geometry is unmeshed</u>

DISCRETIZATION DENSITY

- edge size
- number of divisions

<u>Comment: size can be difined by the size of the element edge or the numer of divisions</u>

DISCRETIZATION TYPE

- Maped
- Sweep
- Free
- Element shape

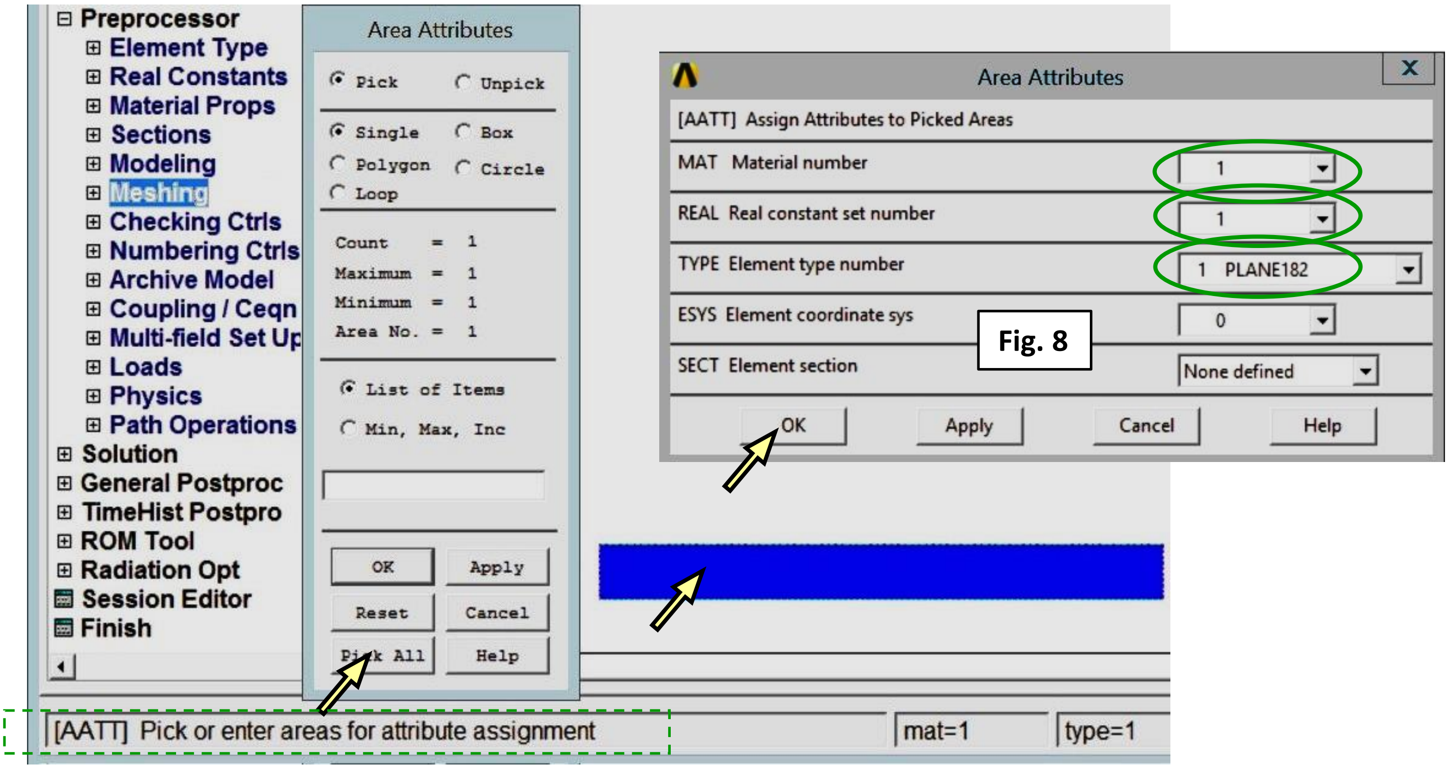
Comment: Mapped and Sweep meshes are recommended, however they are not always easy to create. For example, a mapped mesh on a rectangle requires that the two opposite edges have the same sizes.

Clear mesh to change attributes, a size or the discretization type

Create mesh

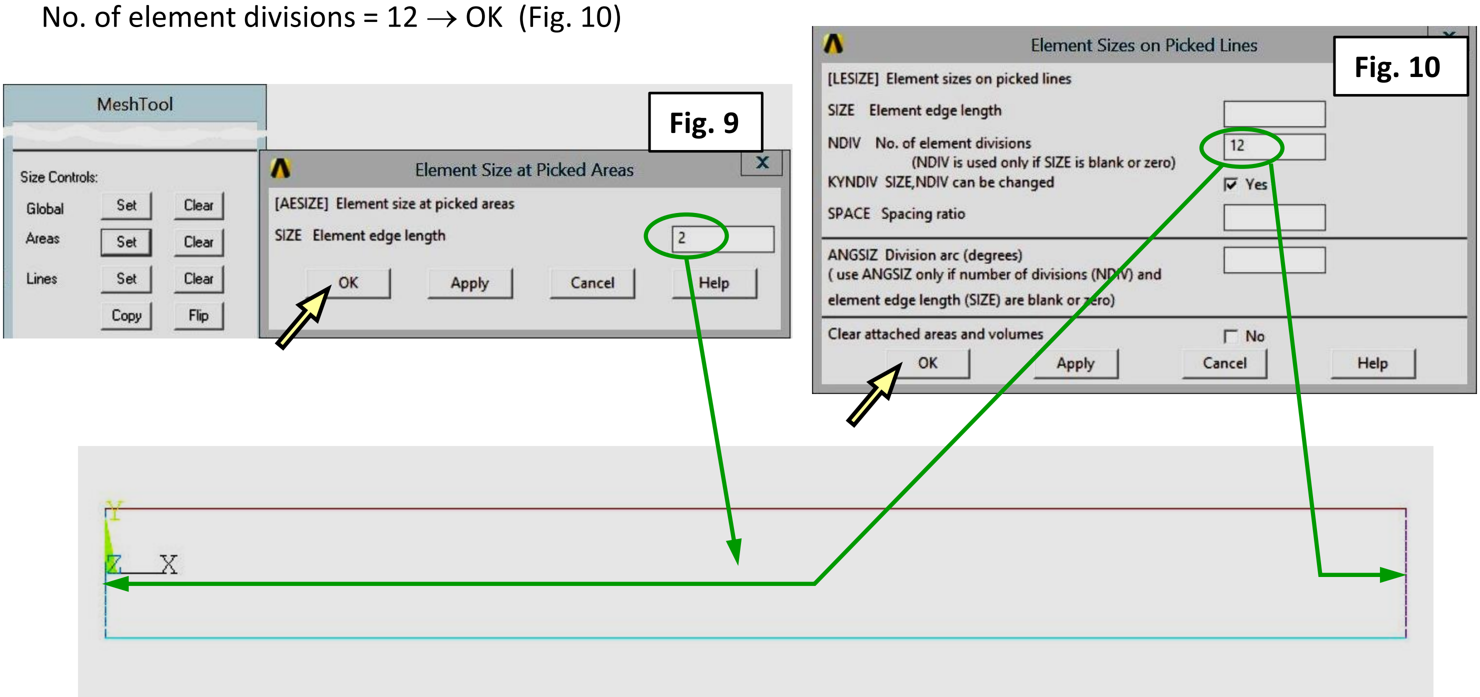
Assign attributes (material, real constant, element type)

Main Menu > Preprocessor > Meshing > Mesh Tool > Element Attributes > Areas → Set (Fig. 7)
Pick the rectangle → OK
Assign attributes: Material numer (1), Real constant set numer (1), Element type number (1) → OK (Fig. 8)



Define discretization density

Main Menu > Preprocessor > Meshing > Mesh Tool > Size Controls > Areas → Set > pick the rectangle → OK
Element edge length = 2 → OK (Fig. 9)
Main Menu > Preprocessor > Meshing > Mesh Tool > Size Controls > Lines → Set > pick vertical lines → OK
No. of element divisions = 12 → OK (Fig. 10)



Define discretization type and mesh

Main Menu > Preprocessor > Meshing > Mesh Tool > Mesh > Areas > Quad > Mapped → Mesh > pick the rectangle → OK (Fig. 11)
Main Menu > Preprocessor > Meshing > Mesh Tool → Close

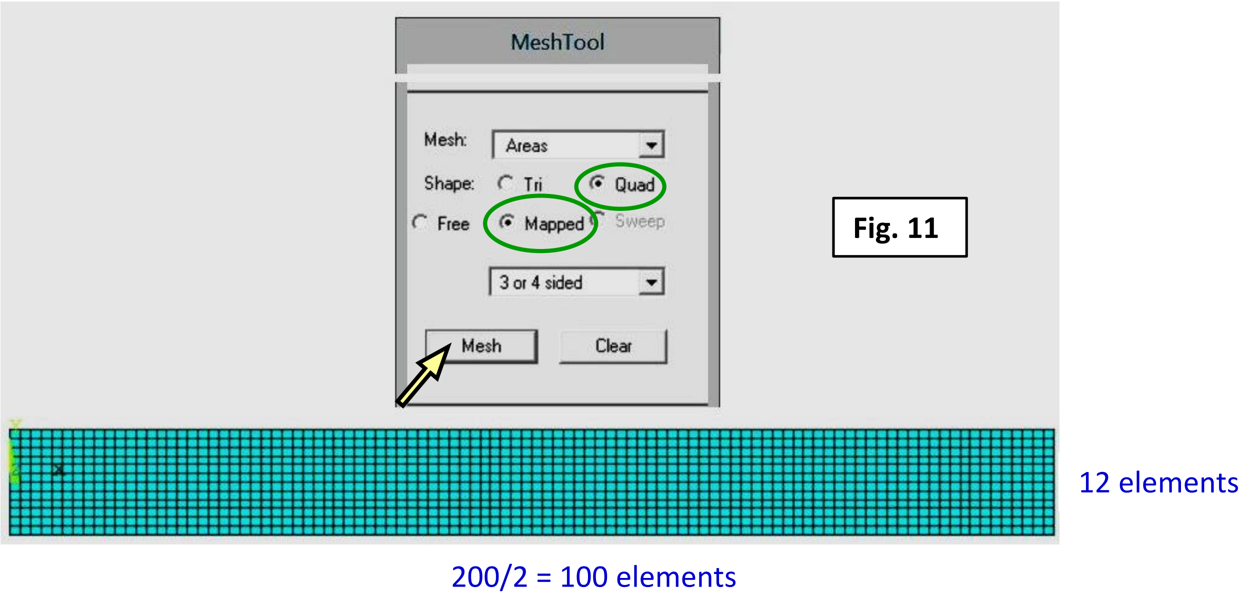


Fig. 11

Save a database file

Utility Menu > File > Save as... beam_FEmodel.db → OK

Define the type of analysis

Main Menu > Solution > Analysis Type > New Analysis > Static → OK (Fig. 12)

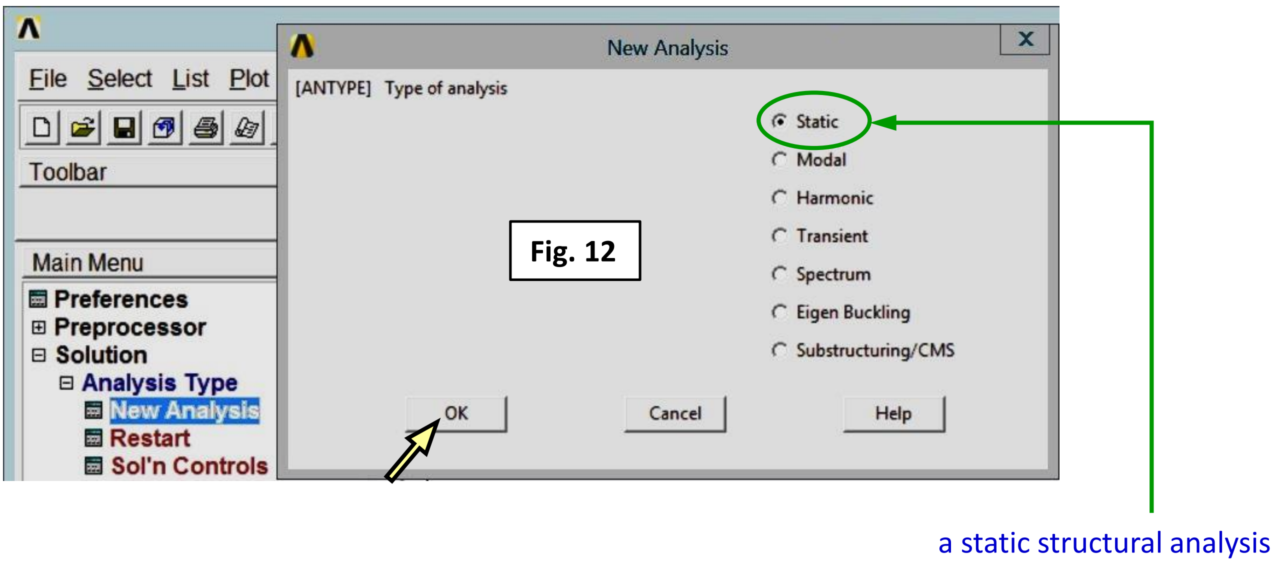


Fig. 12

Define boundary conditions

Support

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Lines >
select the vertical line on the left → OK > DOFs to be constrained > All DOF = 0 → OK (Fig. 13)
Main Menu > Preprocessor > Meshing > Mesh Tool → Close

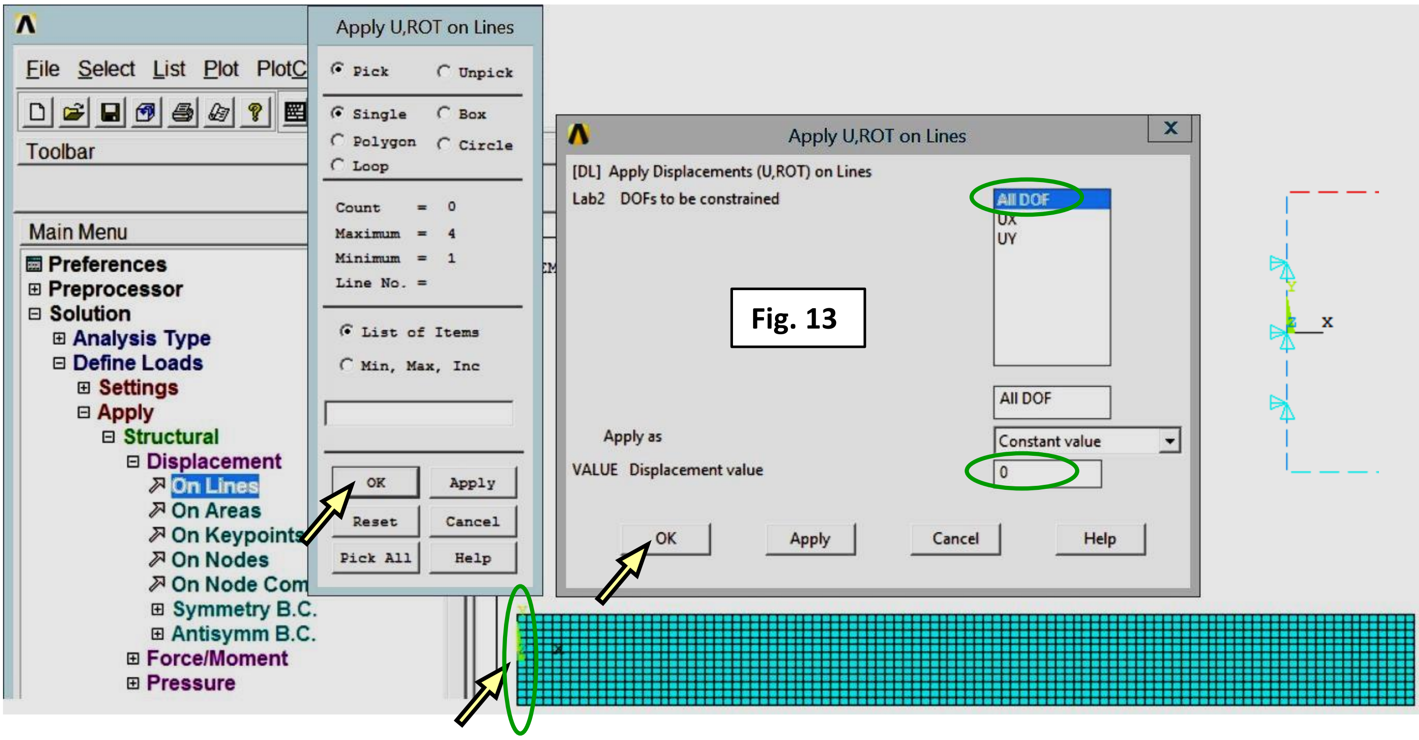


Fig. 13

fixed support

Surface load

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Lines >
select the upper horizontal line → OK > Load PRES value = 2 → OK (Fig. 14)

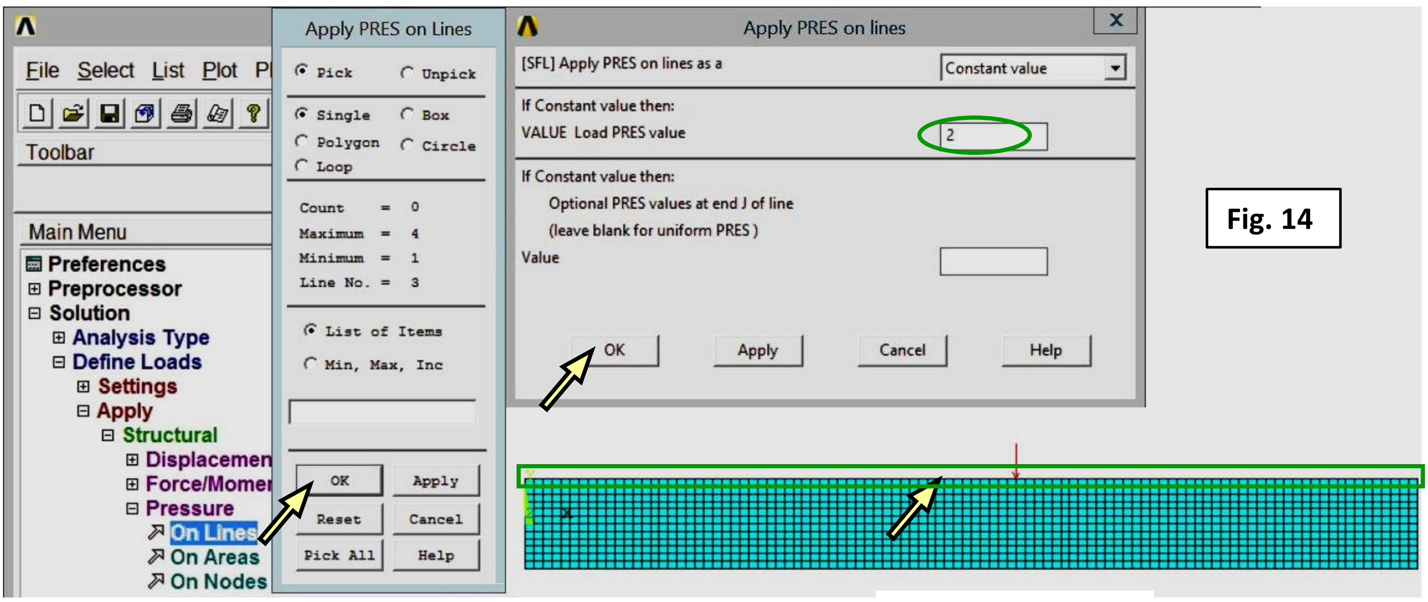


Fig. 14

pressure 6 MPa

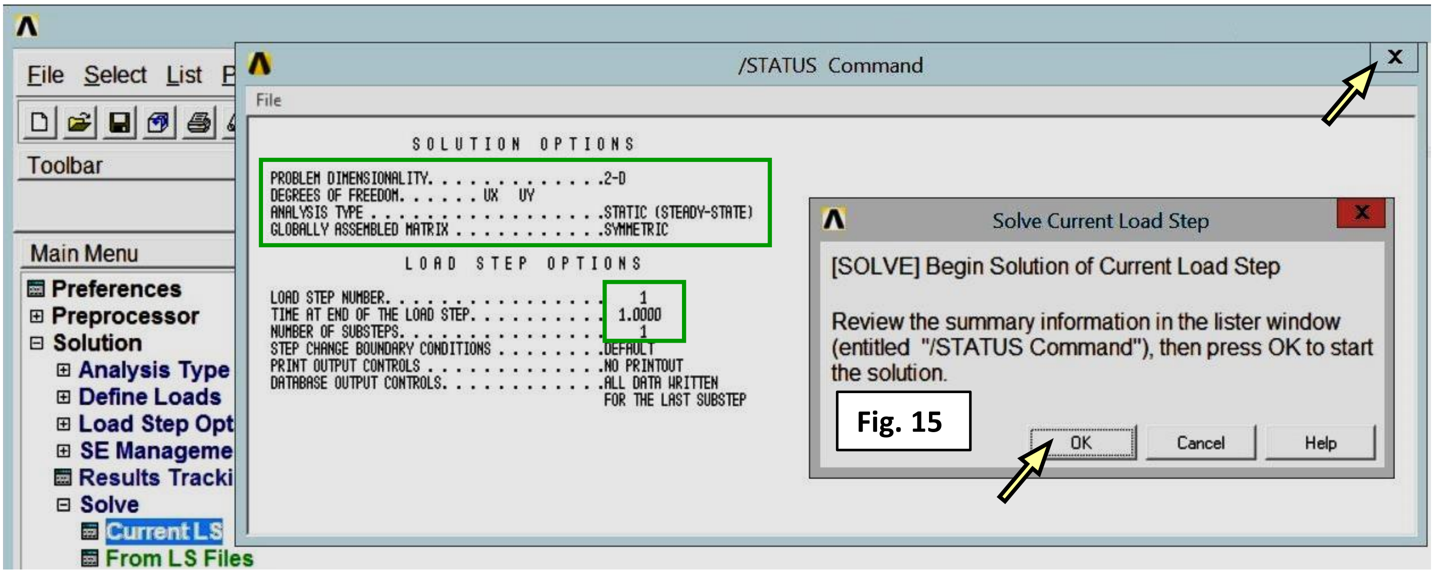
Save a database file

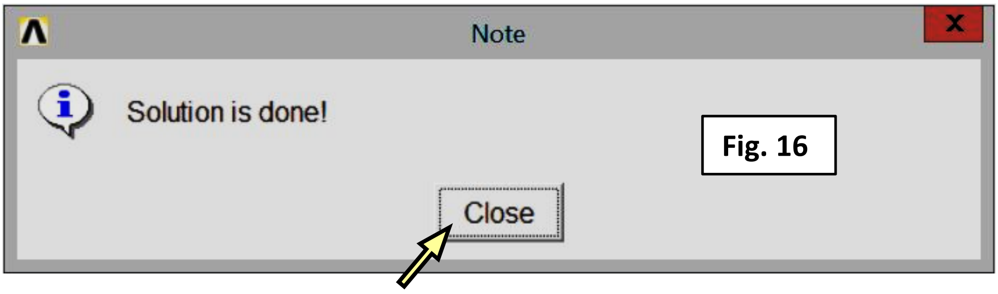
Utility Menu > File > Save as... beam_FEmodel_BC.db → OK

Computation

Solve

Main Menu > Solution > Solve > Current LS > /STATUS COMAND → Close → OK (Fig. 15)
Solution is done! → Close (Fig. 16)





Read the Output Window

```
...
  1   1200 PLANE182   0.000   0.000000
....
SPARSE MATRIX DIRECT SOLVER.
 Number of equations =      2600,   Maximum wavefront =    12
 Memory allocated for solver =        15.259 MB
 Memory required for in-core =          1.488 MB
 Memory required for out-of-core =        0.545 MB
...
 *** NOTE ***                    CP =      80.375   TIME= 18:52:48
 The Sparse Matrix solver is currently running in the in-core memory mode.  This memory mode uses the most amount of
memory in order to avoid using the hard drive as much as possible, which most often results in the fastest solution time.  This
mode is recommended if enough physical memory is present to accommodate all of the solver data.
...
 *** LOAD STEP     1   SUBSTEP     1  COMPLETED.    CUM ITER =      1
 *** TIME =   1.00000         TIME INC =   1.00000      NEW TRIANG MATRIX
 *** NOTE ***                    CP =      81.125   TIME= 18:52:48
 Solution is done!
 *** ANSYS BINARY FILE STATISTICS
  BUFFER SIZE USED= 16384
       0.438 MB WRITTEN ON ASSEMBLED MATRIX FILE: file.full
       1.250 MB WRITTEN ON RESULTS FILE: file.rst
```

number of finite elements included in the analysis

TIME = 1.0 was achieved, so the analysis was succesfully finished

the results file was saved

Save a database file

Utility Menu > File > Save as... beam_FEmodel_results.db → OK (the database with results)

Results

Contour map of the displacement in y direction

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > DOF Solution >
> Y – Component of displacement → OK (Fig. 17)

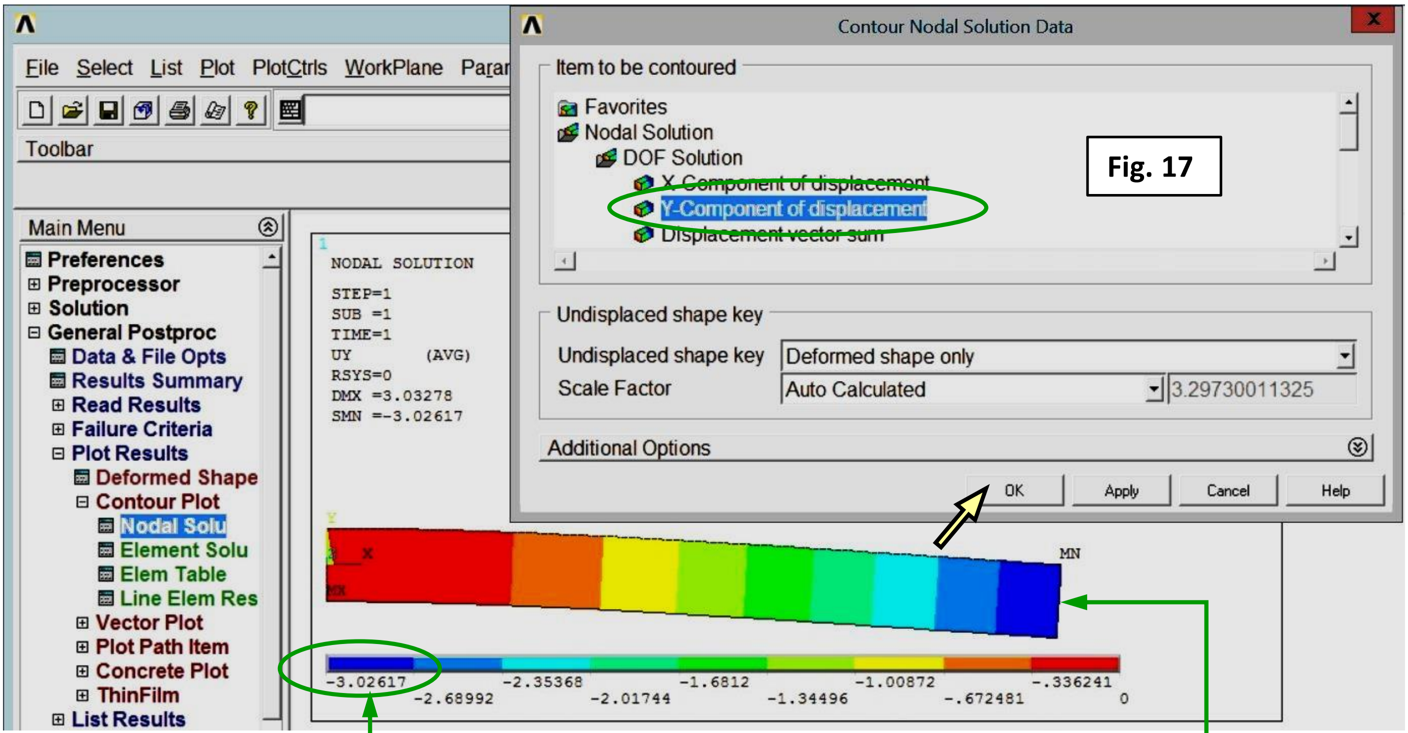


Fig. 17

analytical solution $v(l) = -\frac{p_0 \cdot l^4}{8 \cdot E \cdot J_z} = -3\ \text{mm}$

a relative error $\Delta v(l) = \frac{3.02617-3}{3} = 0.9\%$

Contour map of the normal stress in x direction

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu > Stress >
> X – Component of stress → OK (Fig. 18)

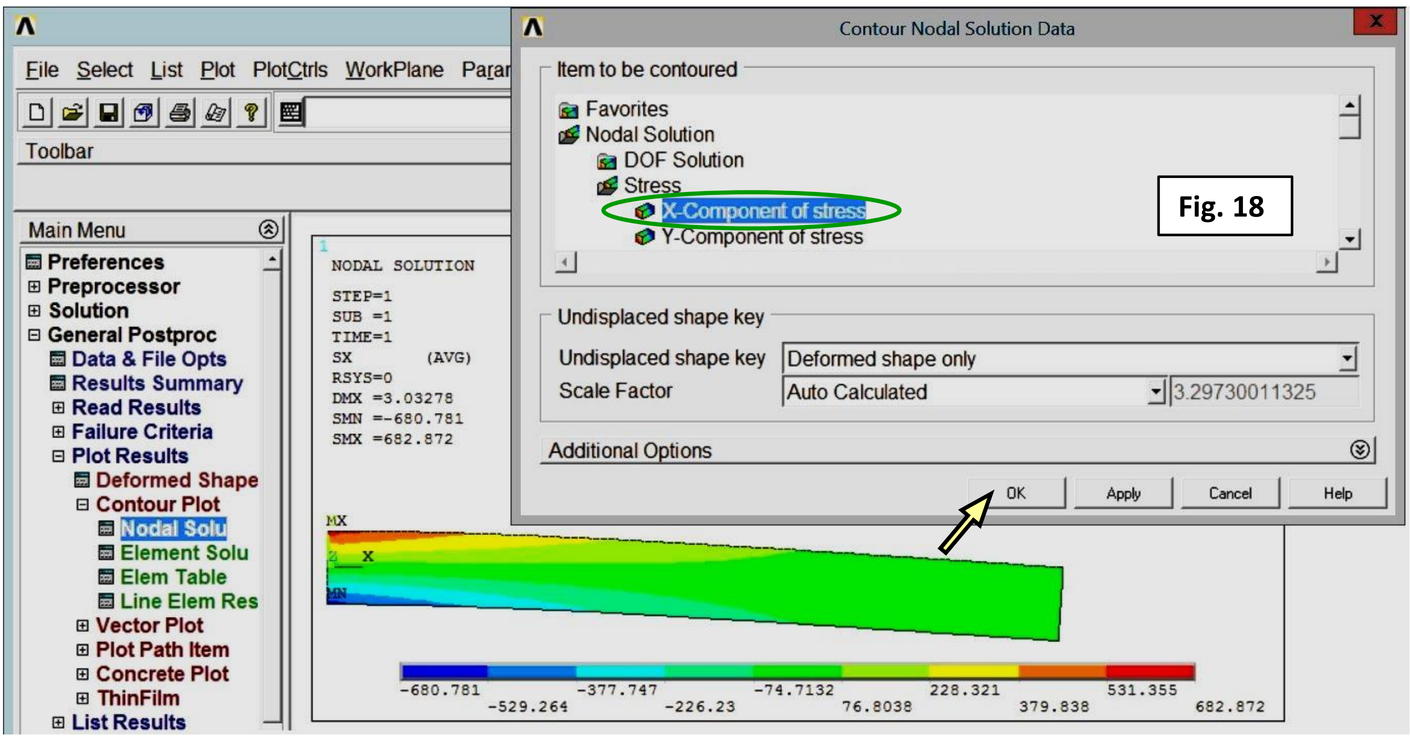


Fig. 18

Stress components in a cross section for x = 100 mm

Select nodes in the cross section x = 100 mm

Utility Menu > Select > Entities … > Nodes > By Location > X coordinates >
> Min = 99, Max = 101 > From Full→ OK (Fig. 19)
Utility Menu > Plot > Nodes

Define a path

Main Menu > General Postproc > Path Operations > Define Path > By Nodes
pick the start and end nodes → OK (Fig. 20)
Define a path name: Path_1 → OK (Fig. 21)

Map stress components

Main Menu > General Postproc > Path Operations > Map onto Path >
Stress: SX → Apply > shear SXY → Apply > von Mises SEQV → OK (Fig. 22)

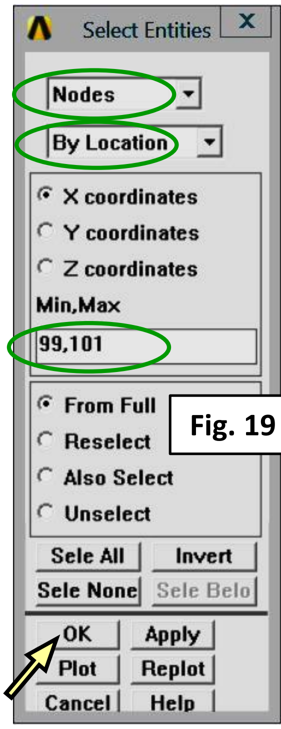


Fig. 19

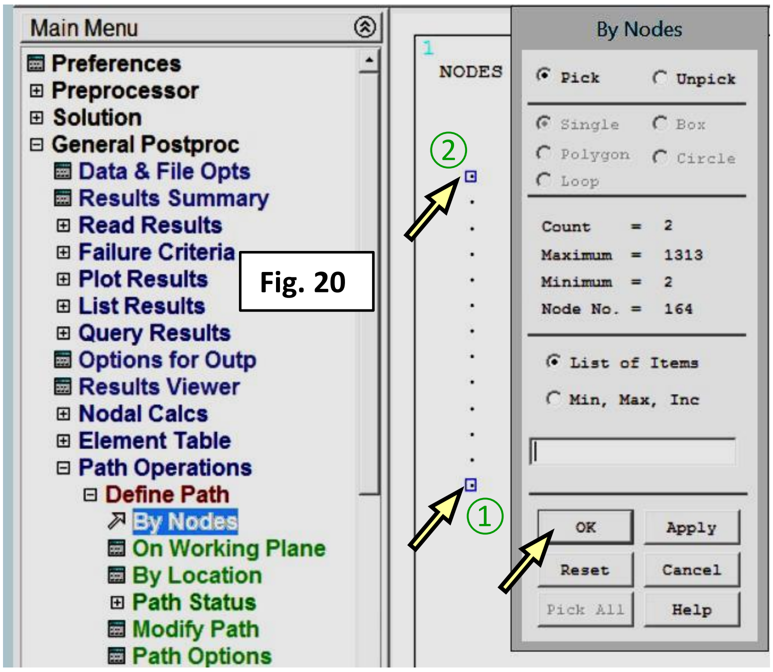


Fig. 20

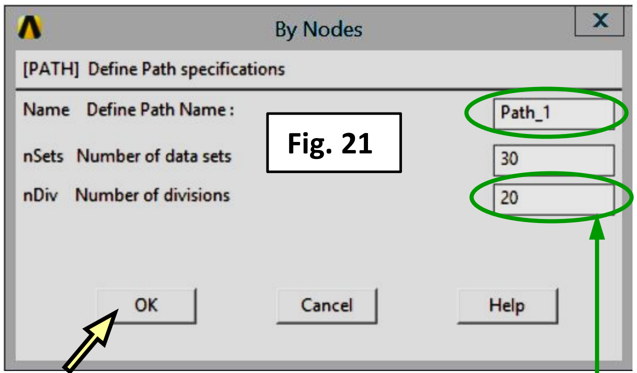


Fig. 21

For nDiv= 20 the graph is based on 21 points between the start and end nodes

the field 'Lab' can be left blank

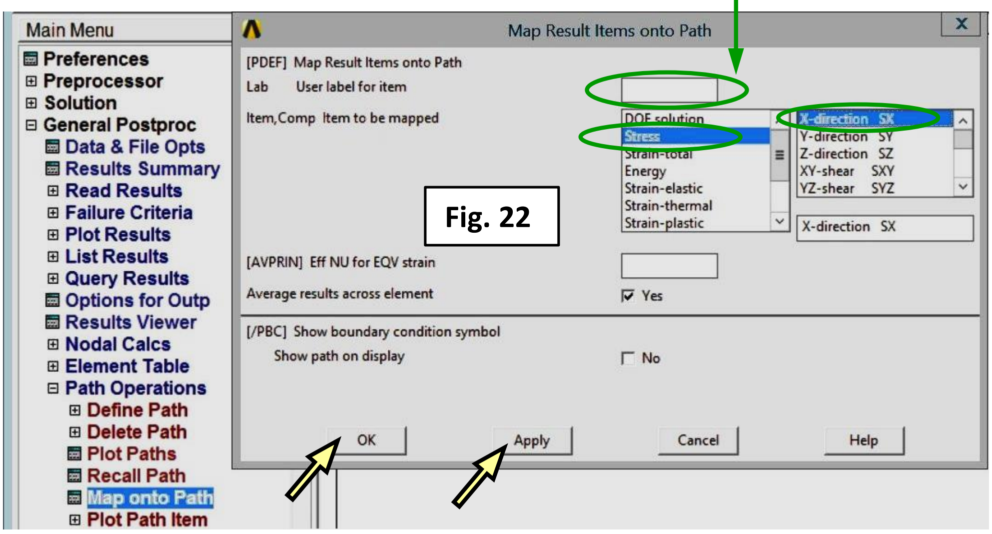


Fig. 22

Plot stress components on a graph

Main Menu > General Postproc >
Path Operations > Plot Path Item > On Graph
SX, SXY, SEQV → OK (Fig. 22, 23)

Plot a shear stress on a graph

Main Menu > General Postproc >
Path Operations > Plot Path Item > On Graph
SXY → OK
PlotCtrls > Style > Graphs > Modify Axes … >
Y-axis range > Specified range :
YMIN = -15, YMAX = 0 → OK
Utility Menu > Plot > Replot (Fig. 24, 25)

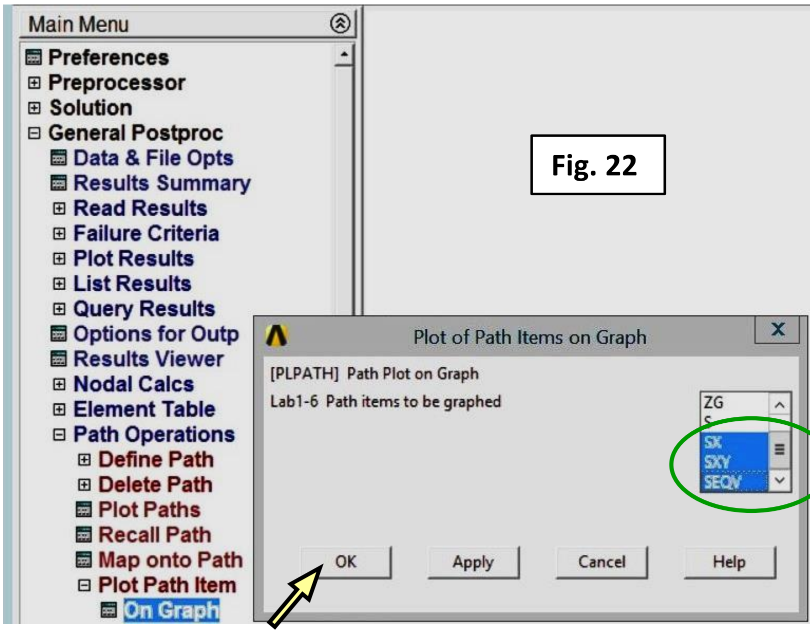


Fig. 22

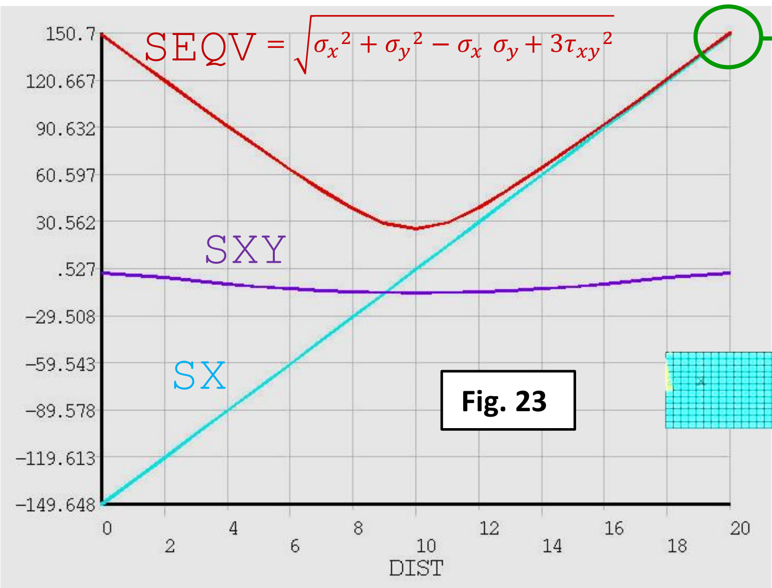
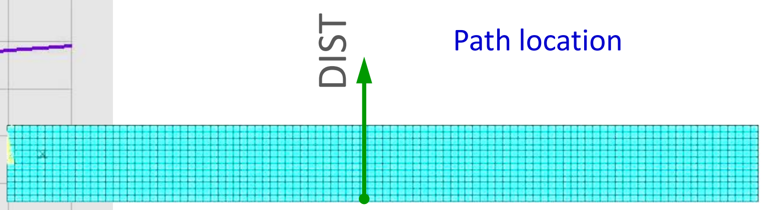


Fig. 23

analytical solution: $\sigma_x\left(\frac{l}{2},\frac{h}{2}\right) = 150$ MPa

relative error: $\Delta\sigma_x\left(\frac{l}{2},\frac{h}{2}\right) = \frac{149.648-150}{150} = -0.2\%$



$$x = \frac{l}{2} = 100 \text{ mm}$$

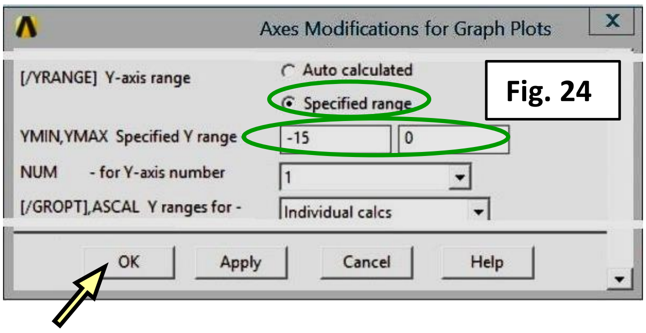


Fig. 24

analytical solution:

$$\tau_{xy}\left(\frac{l}{2},0\right) = -15 \text{ MPa} \ ; \ \tau_{xy}\left(\frac{l}{2},\pm\frac{h}{2}\right) = 0 \text{ MPa}$$

relative error: $\Delta\tau_{xy}\left(\frac{l}{2},0\right) = \frac{-14.861-(-15)}{-15} = 0.9\%$

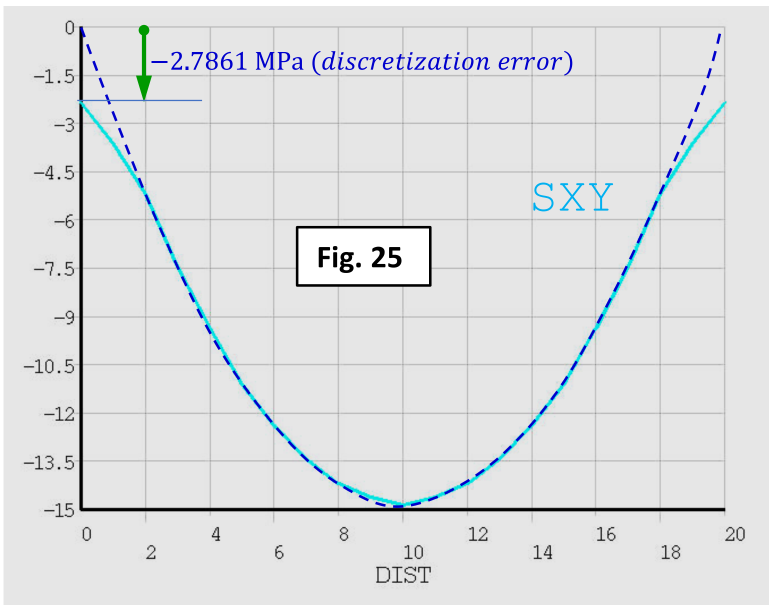


Fig. 25

Select the entire model

Utility Menu > Select > Everything
Utility Menu > Plot > Replot

List reactions

Main Menu > General Postproc > List Results > Reaction Solu > All Items → OK (Fig. 26)

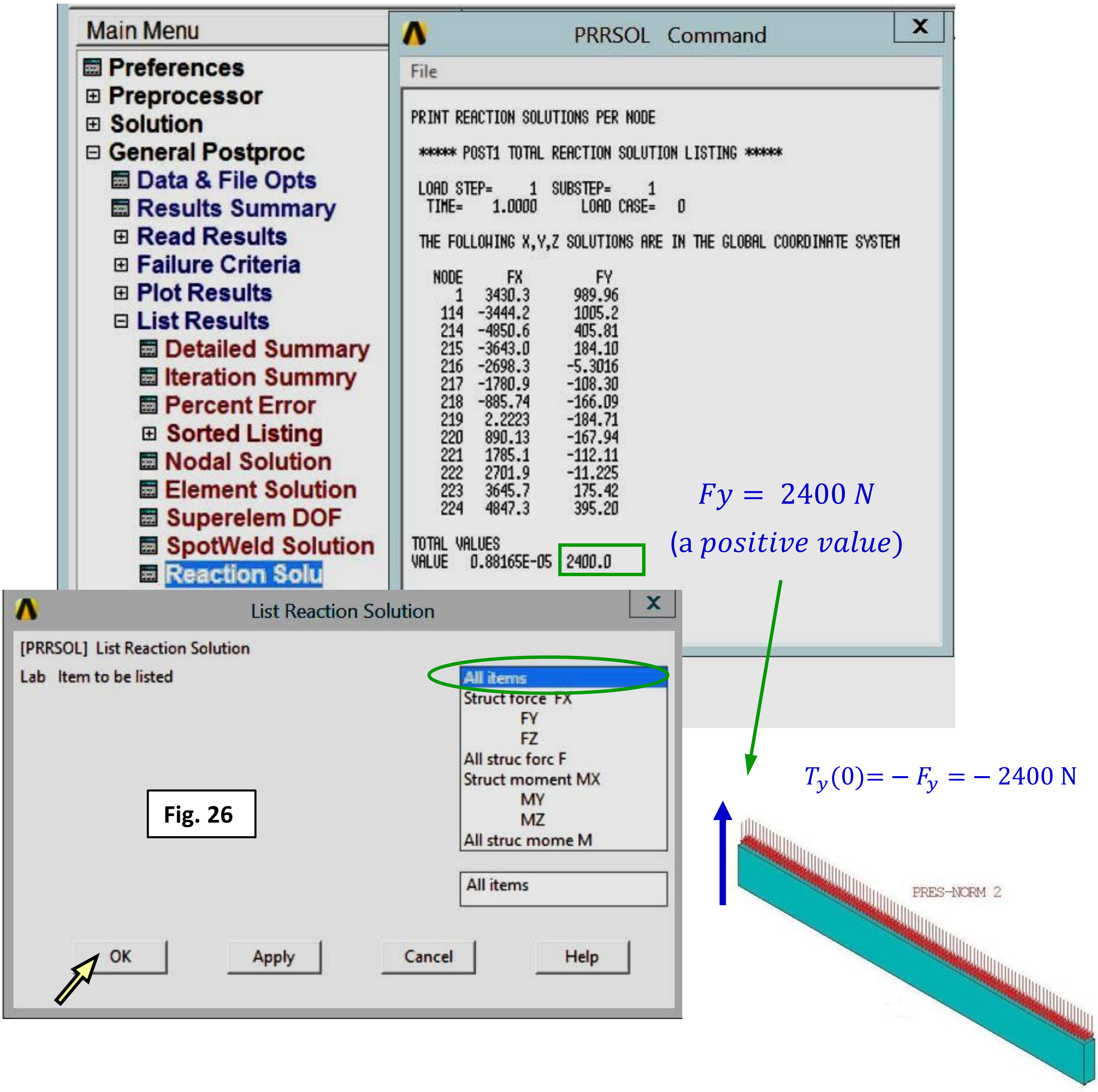


Fig. 26