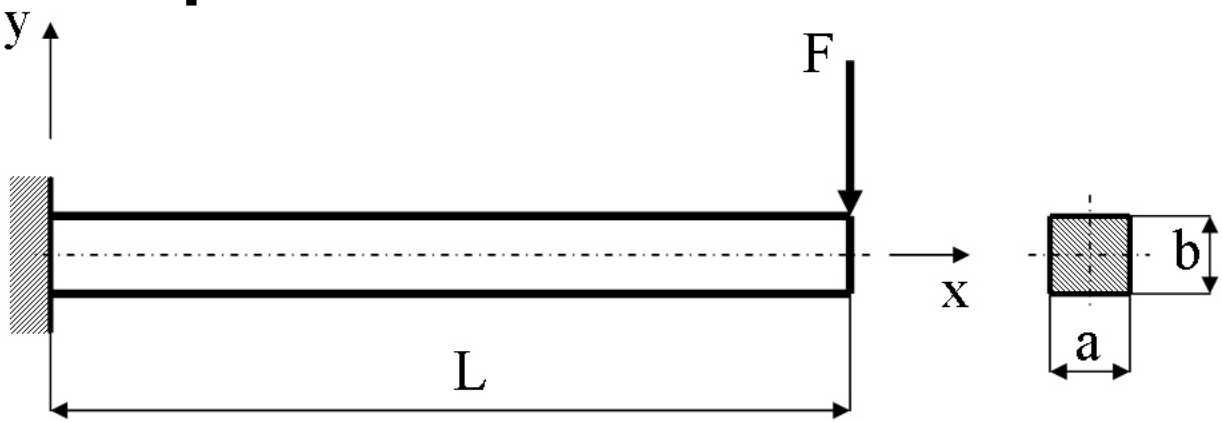
Course in ANSYS

Example0570

Example – Cantilever beam



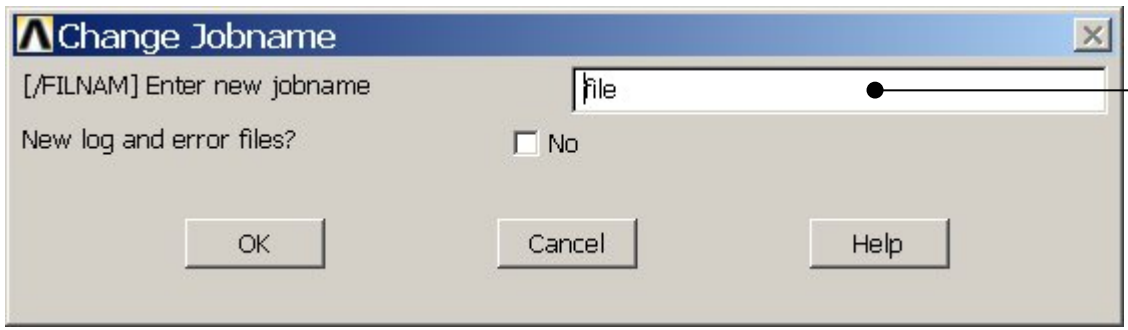
Objective:
Run the problem using different material models
Tasks:
Run a static linear model
Run a static full nonlinear model with:
- A bilinear kinematic hardening behaviour
Topics:
Element type, Real constants, modeling,
Plot results, output graphics, nonlinear solution control

$E = 210000 N/mm^2$
$\nu = 0.3$
$L = 100mm$
$a = 10mm$
$b = 10mm$
$\sigma_y = 355 N/mm^2$
$F = 1000N$

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0570 — Command line entry

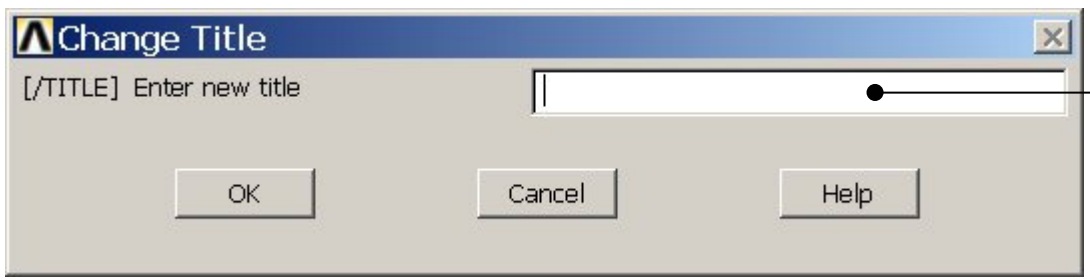


Enter: Example0570

Utility Menu > File > Change Title

/title, Cantilever beam

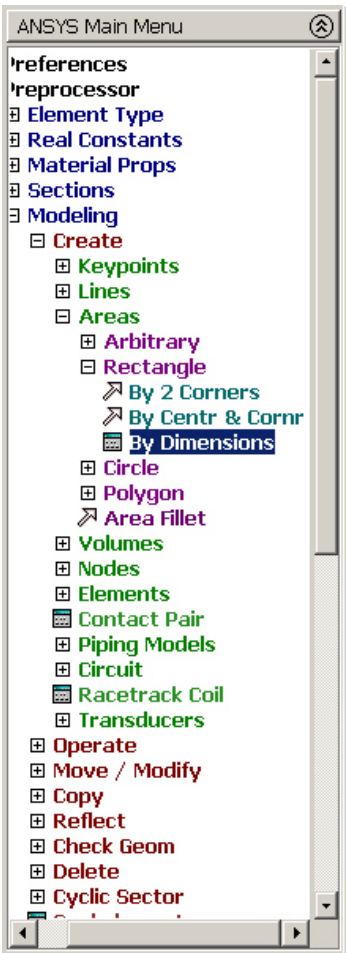
Enter: Cantilever beam

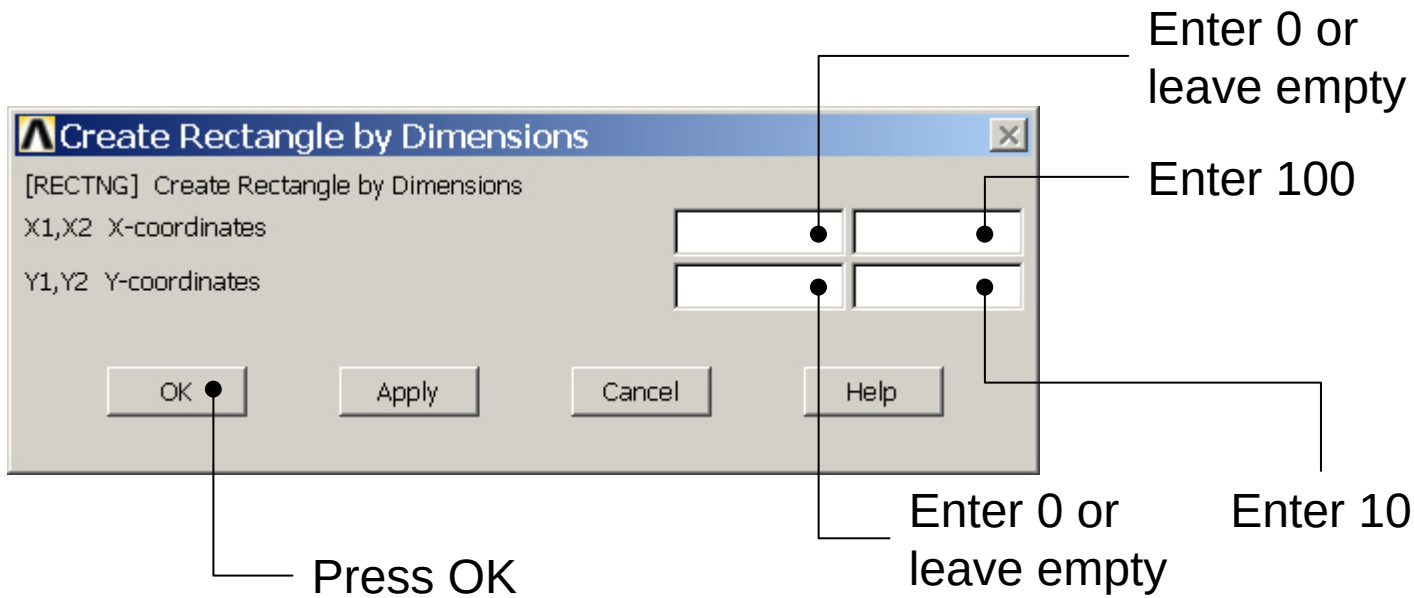


Example - Areas

Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

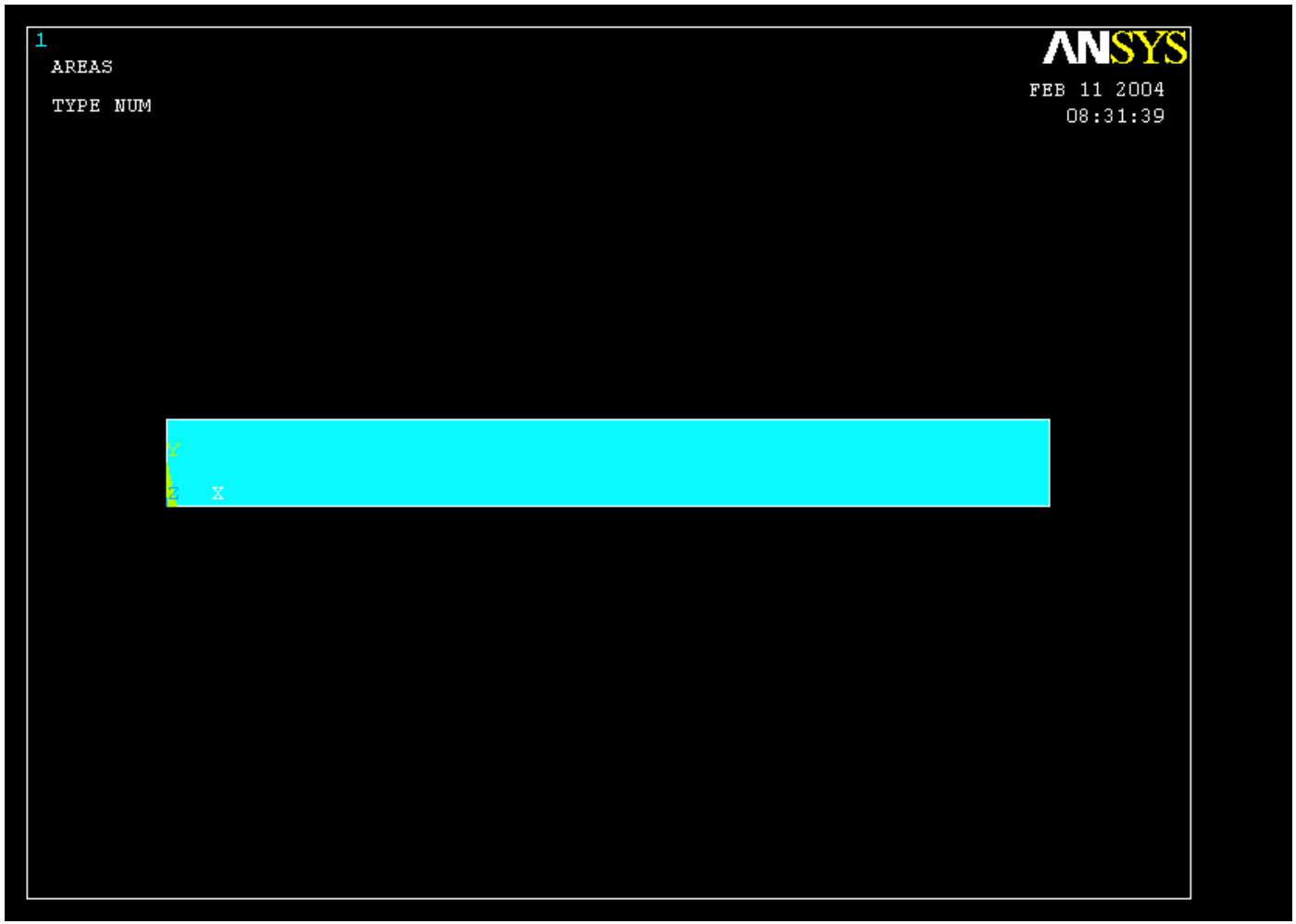
Create an area given by X=(0,100) and Y=(0,10)





Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

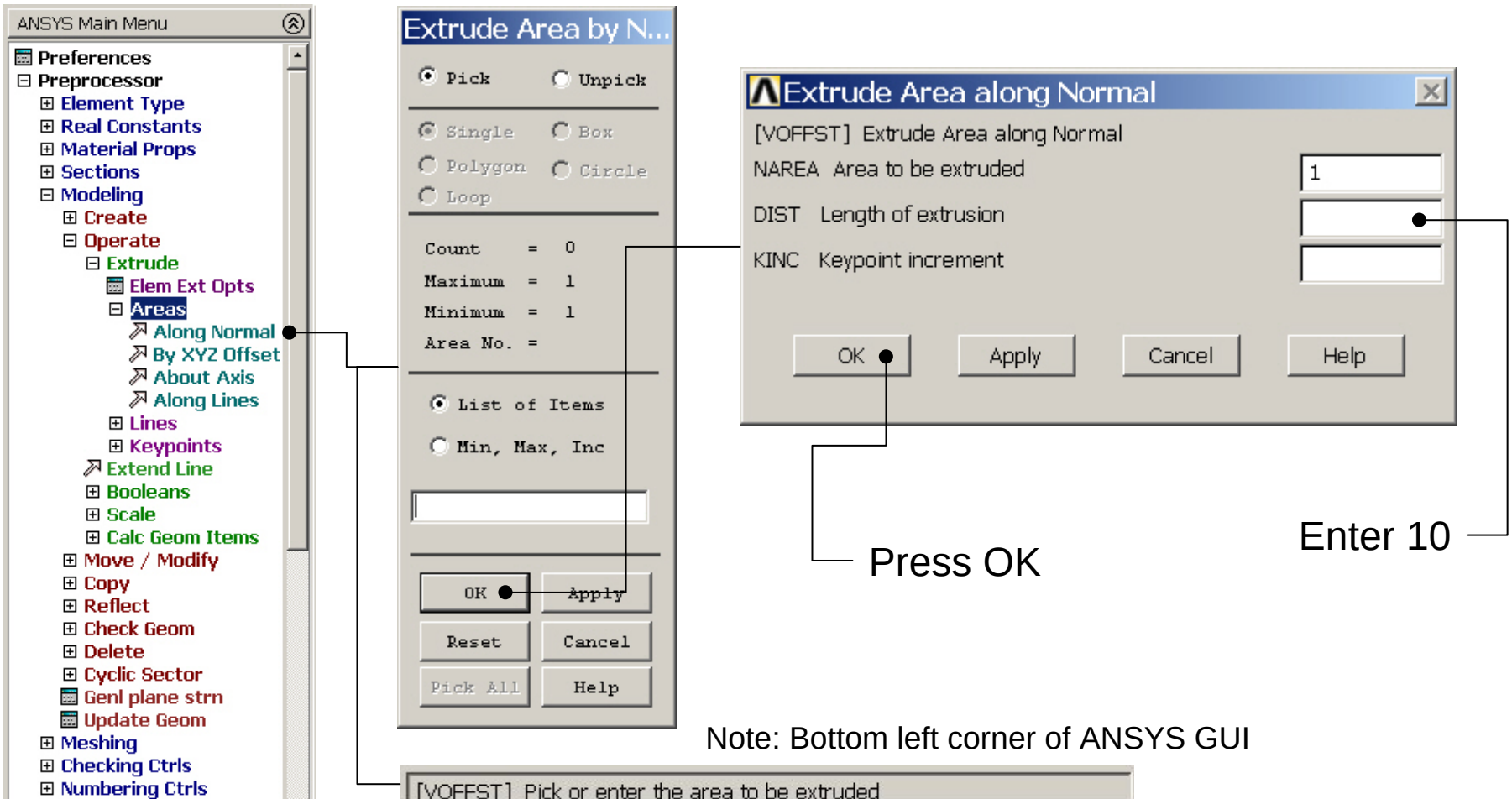
Example - Area



Example - Operate

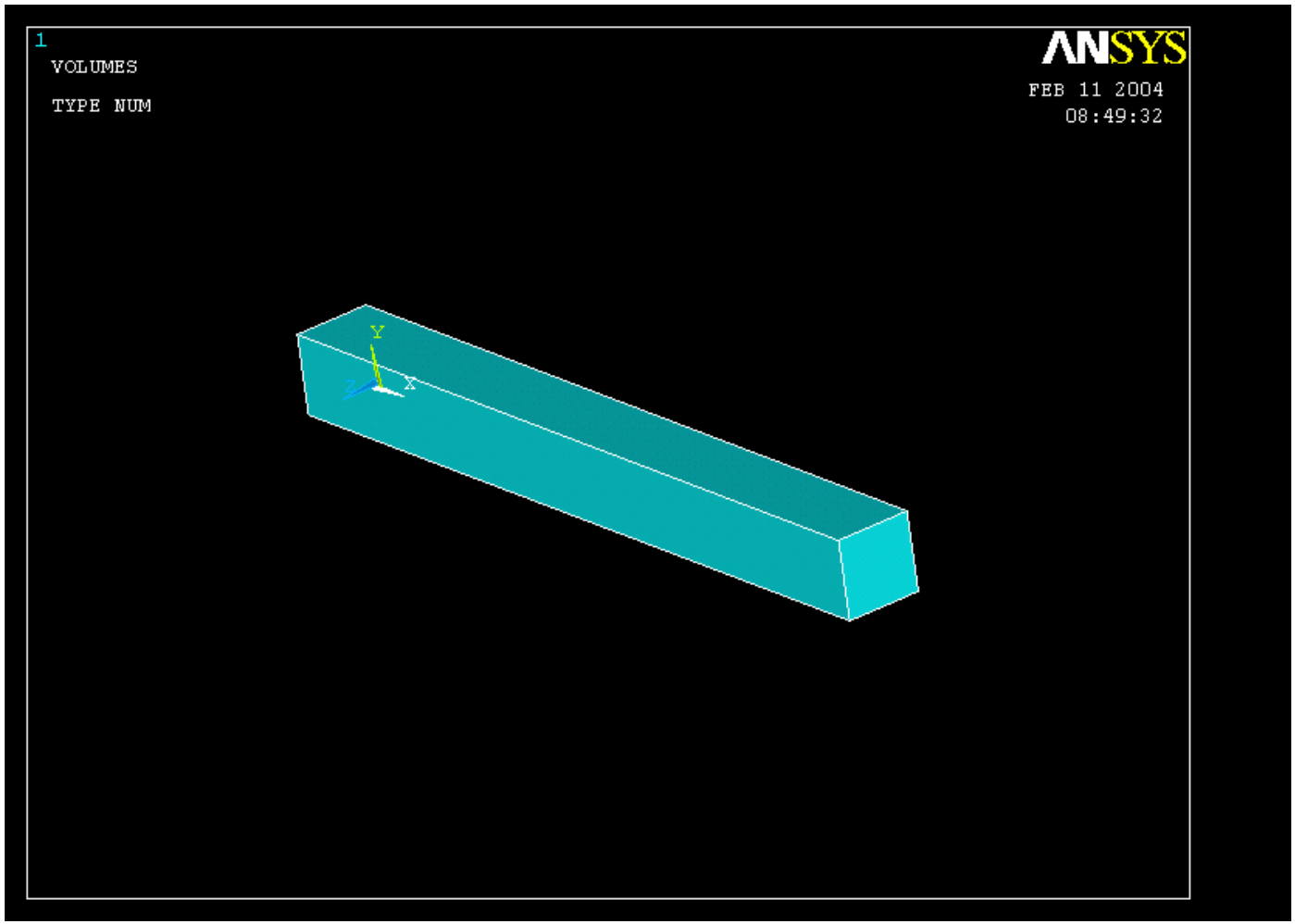
Preprocessor > Modeling > Operate > Extrude > Areas > Along Normal

Create a volume by extruding the area 10 along its surface normal vector



Note: Bottom left corner of ANSYS GUI

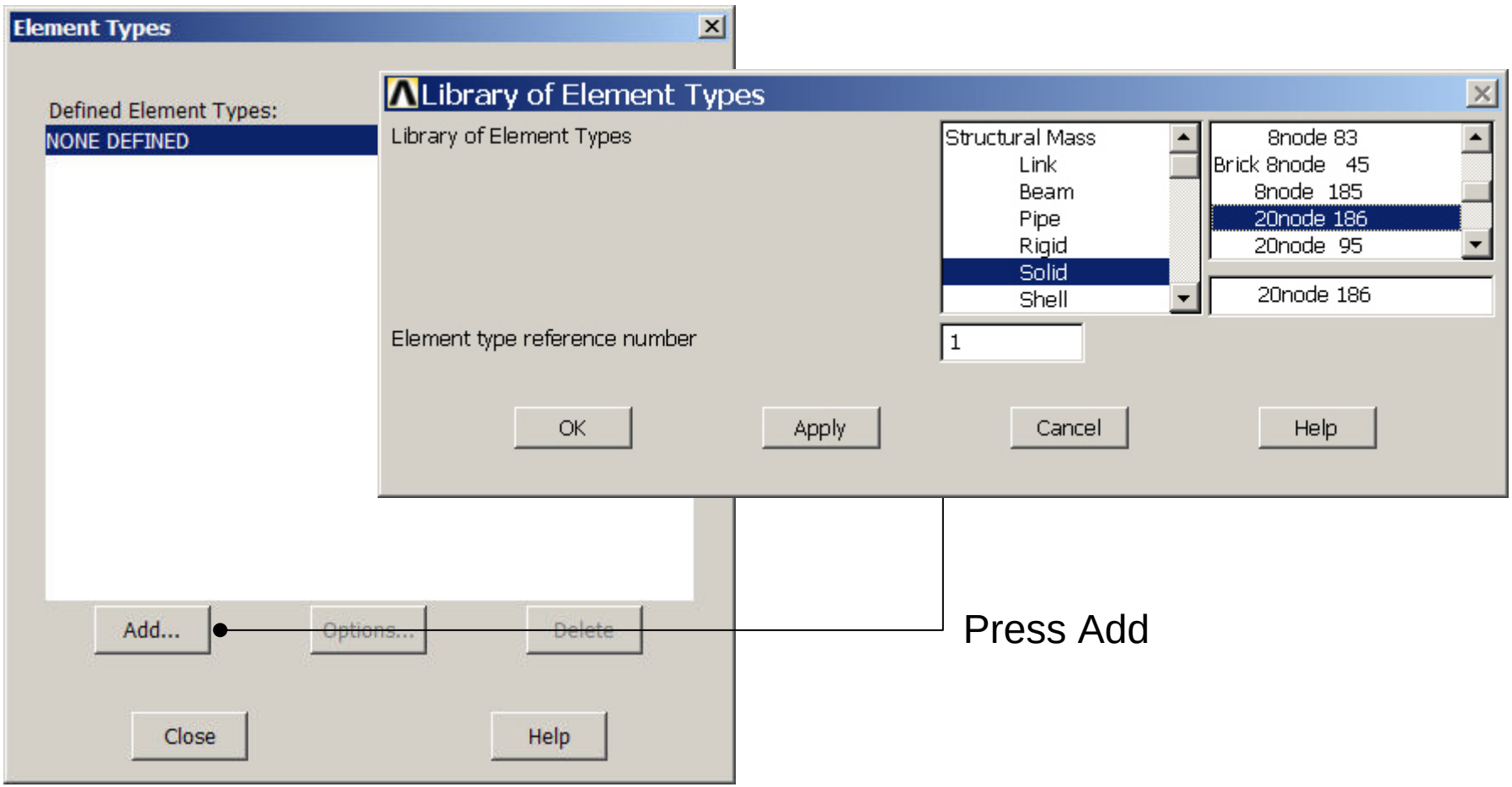
Example – Mouse rotate



Rotate by holding the Ctrl key down while using the right hand mouse button

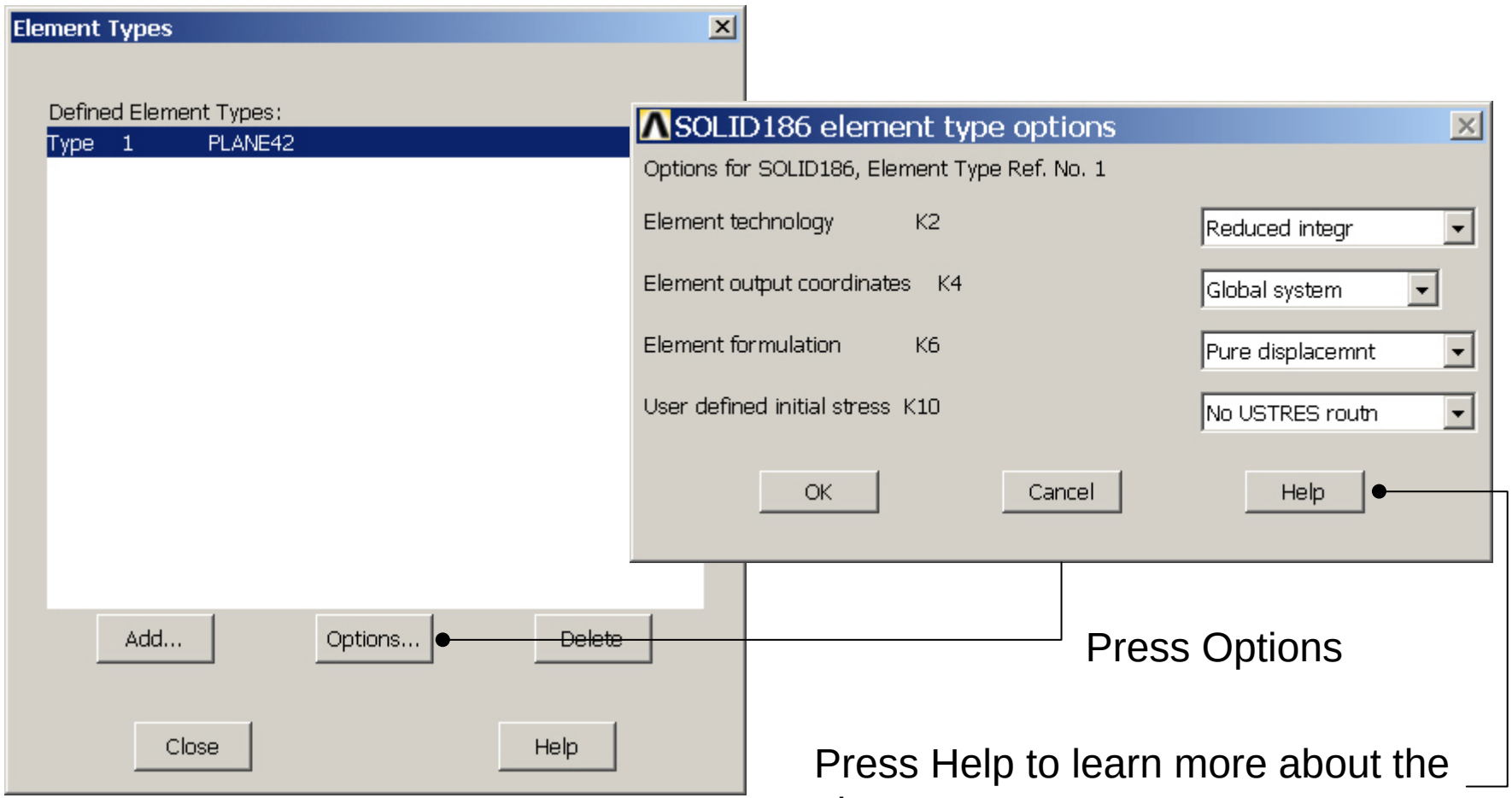
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Press Options

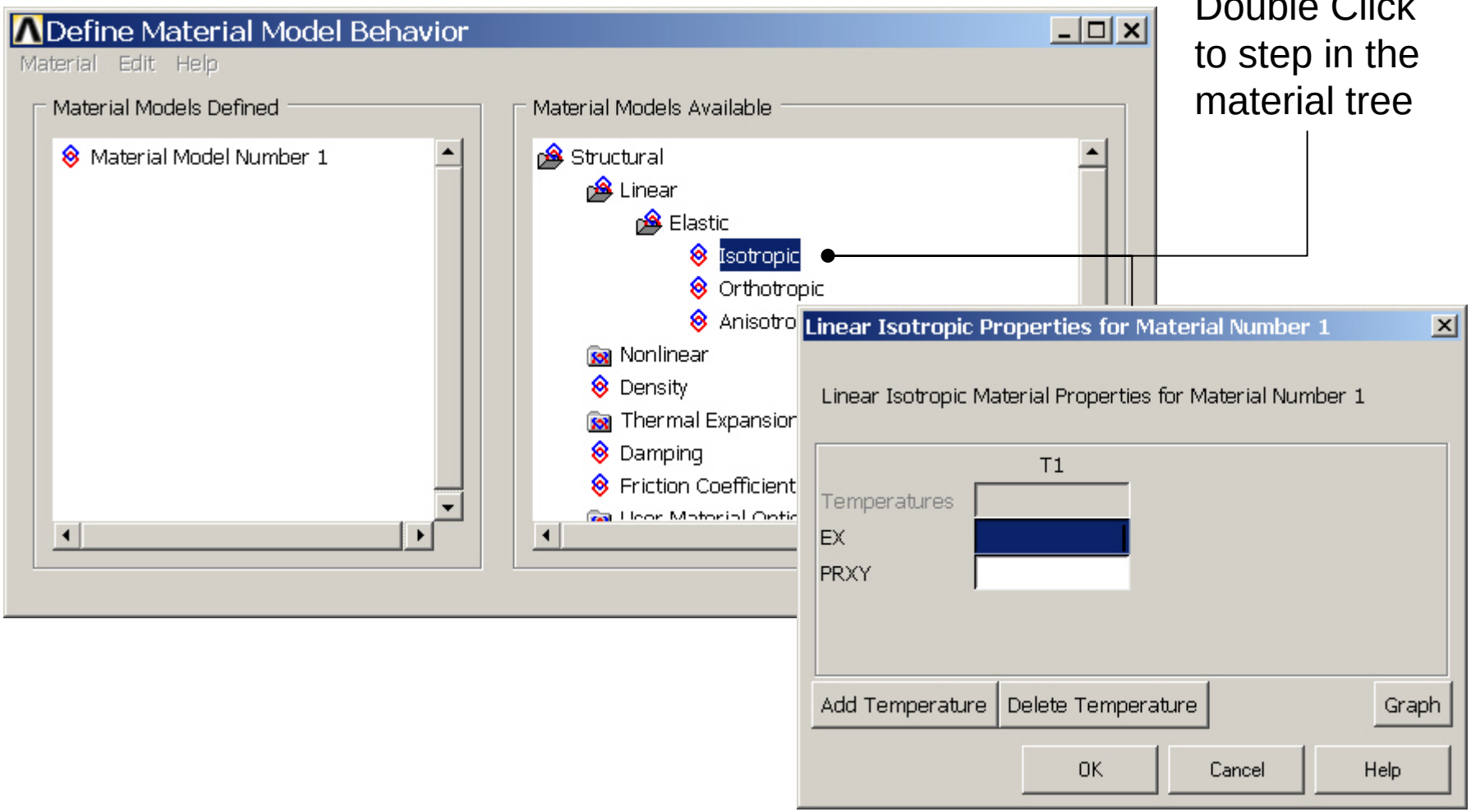
Press Help to learn more about the element.

Example – Real Constants

No Real Constants are necessary for pure 3D solid models!

Example - Material Properties

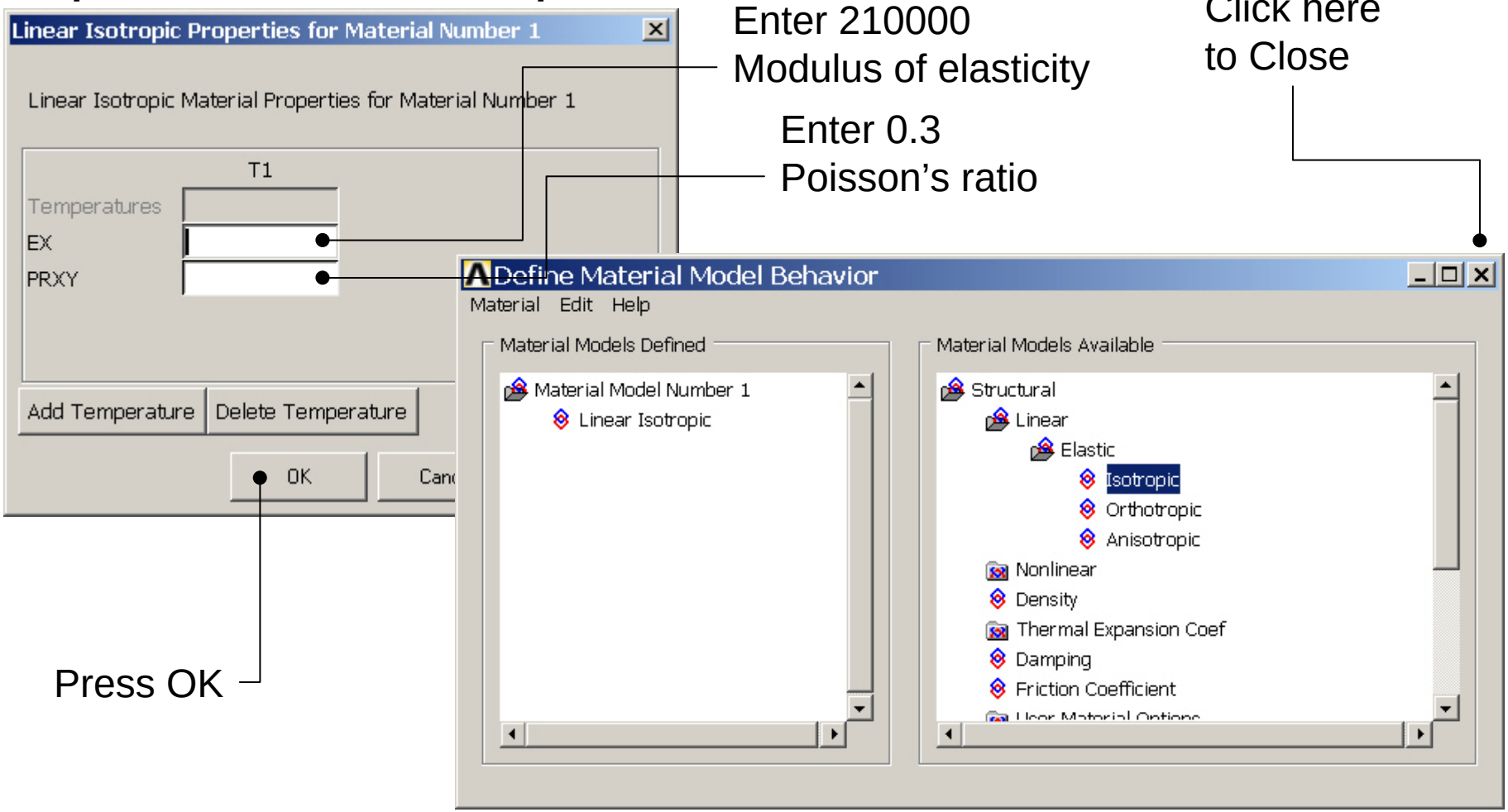
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

Example - Material Properties

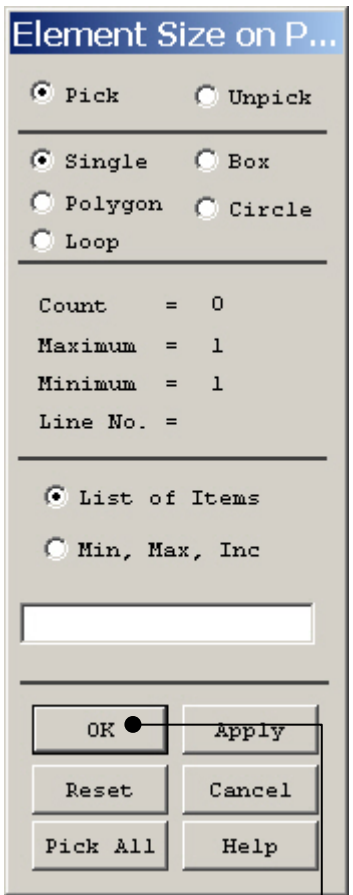
Preprocessor > Material Props > Material Models

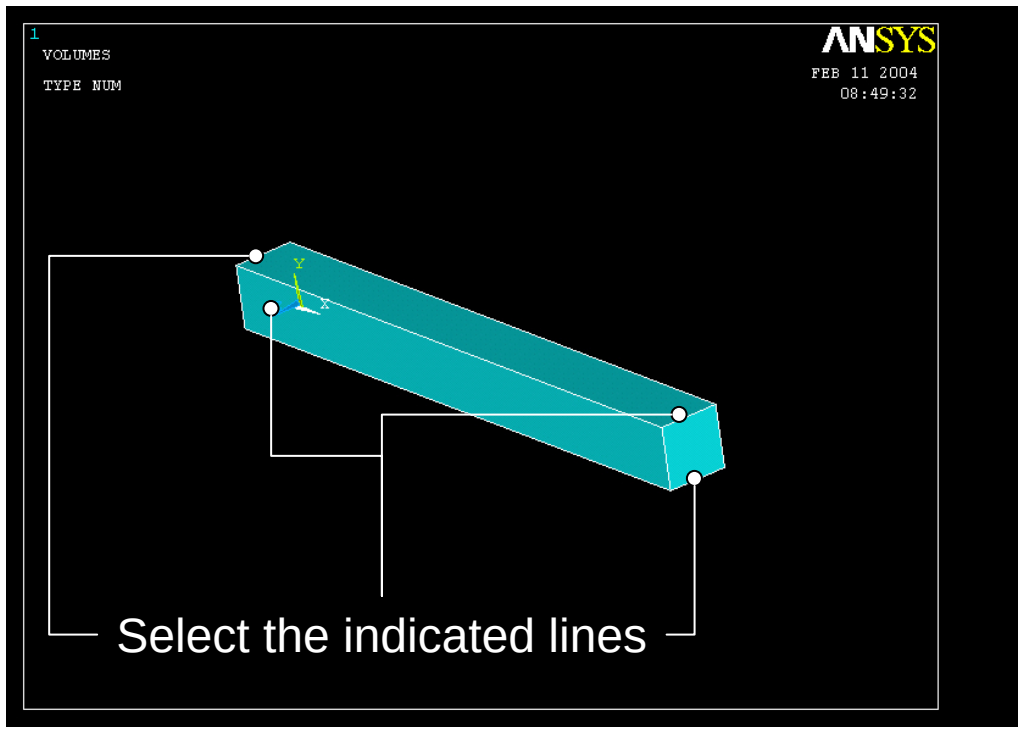


Example - Meshing

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

Select/Pick Lines to specify mesh size for

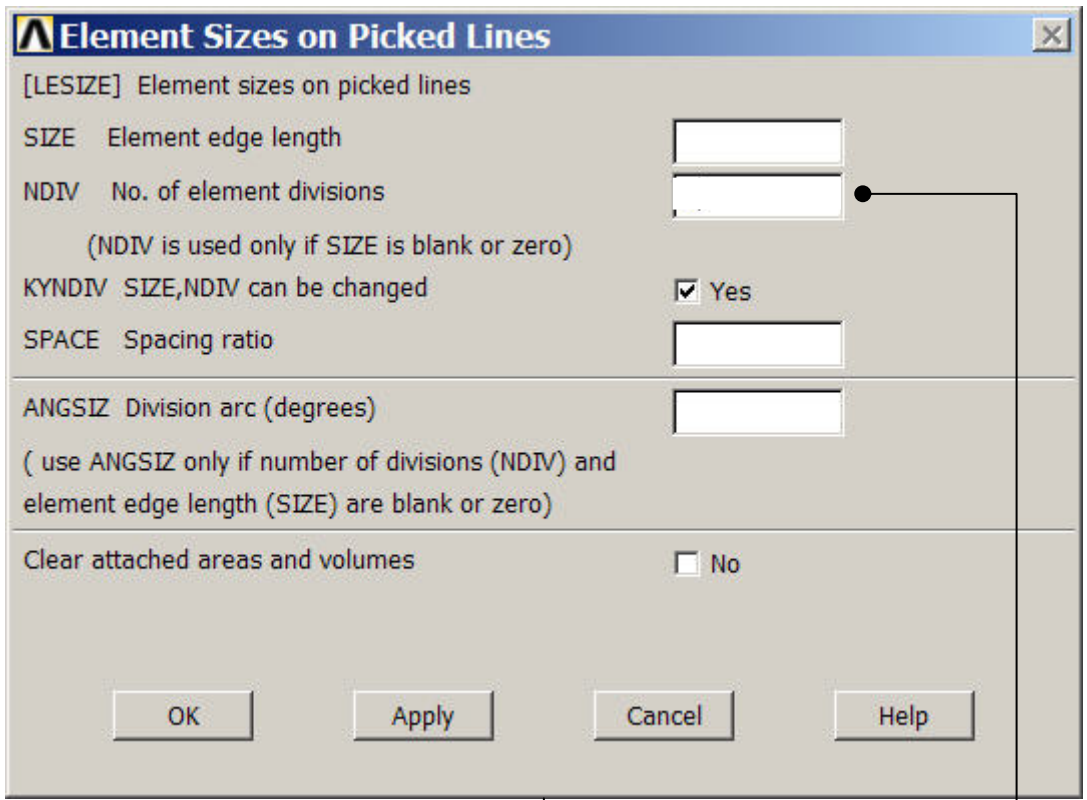




Press OK when finish with selection

Example - Meshing

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

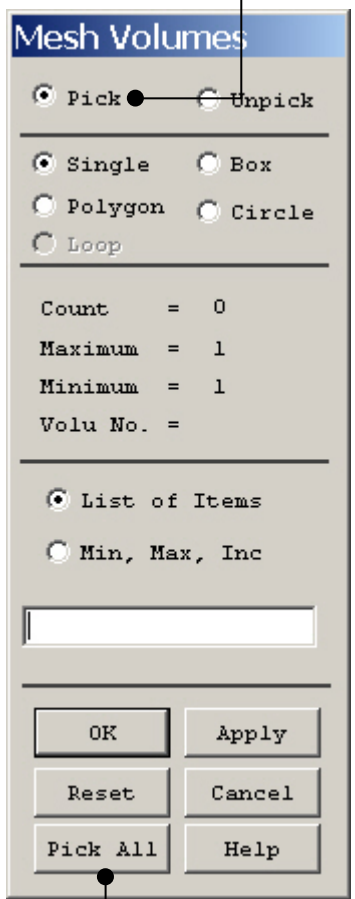


Enter 2

Press OK when finish with selection

Example - Meshing

Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 or 6 sided

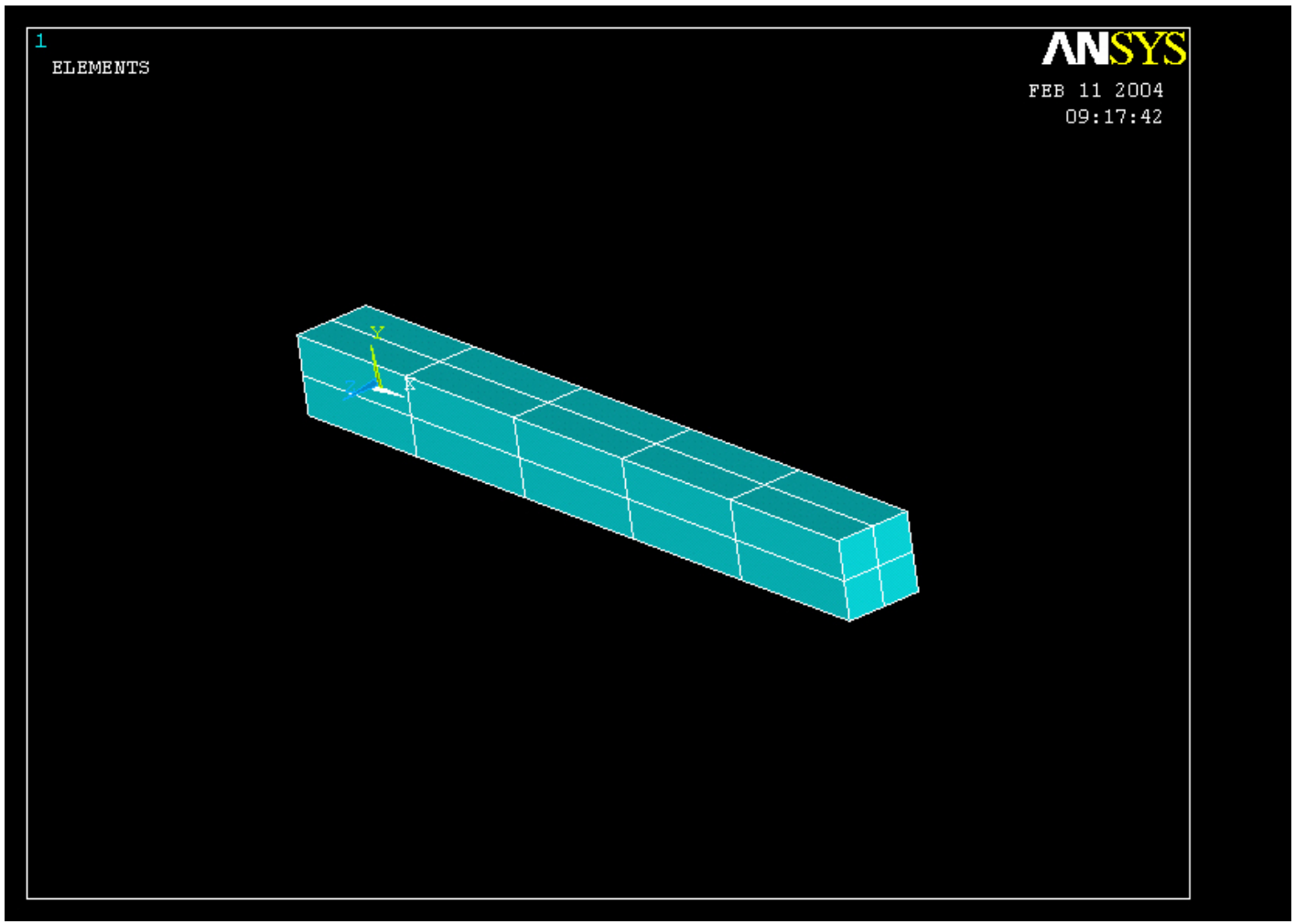


Select individual volumes to be meshed

NB: It is often necessary to “Clear” the model for example if Element Type or model geometry is to be changed

Select all volumes defined to be meshed

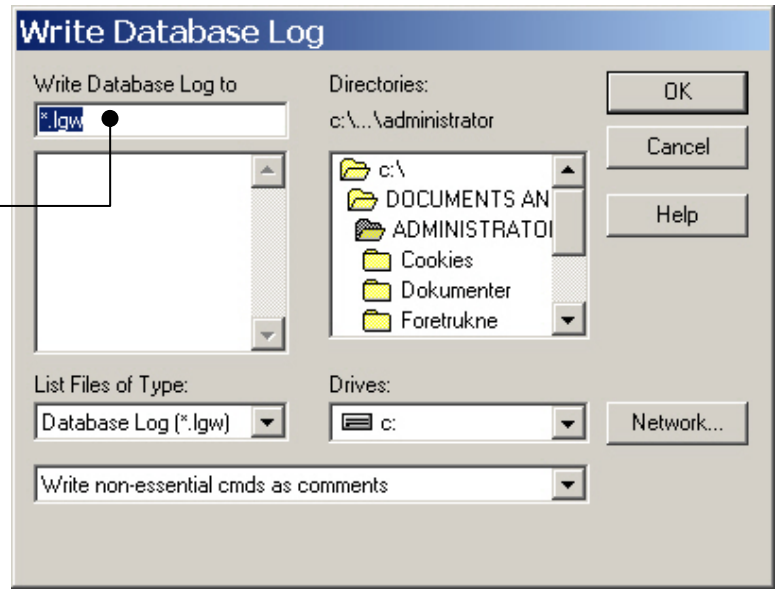
Example – 3D Mesh



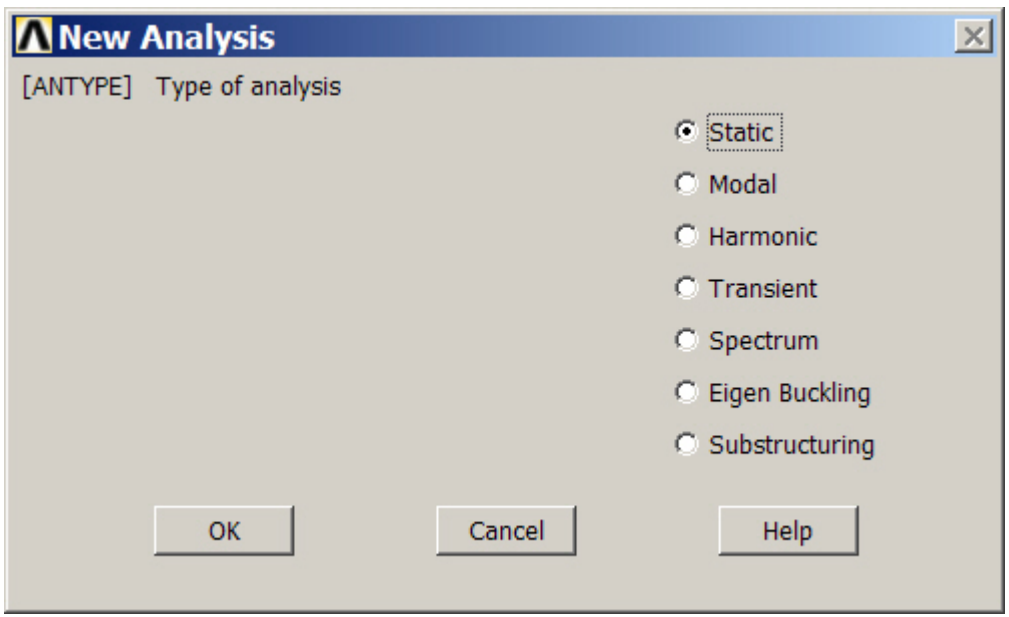
Example – Analysis Type

File > Write DB log file

Enter “example0570.lgw”

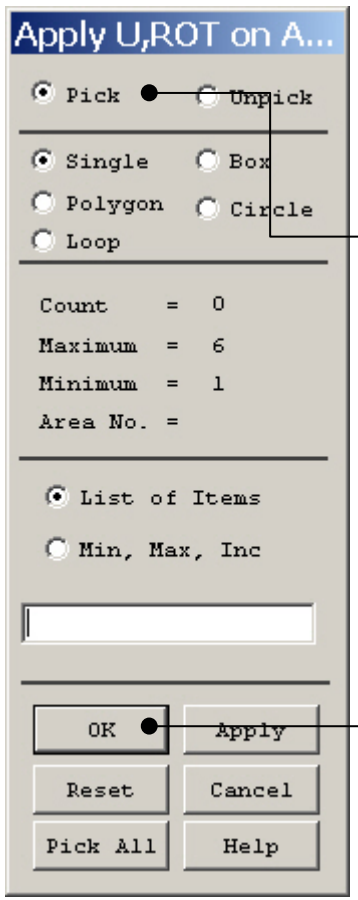


Solution > Analysis Type > New Analysis



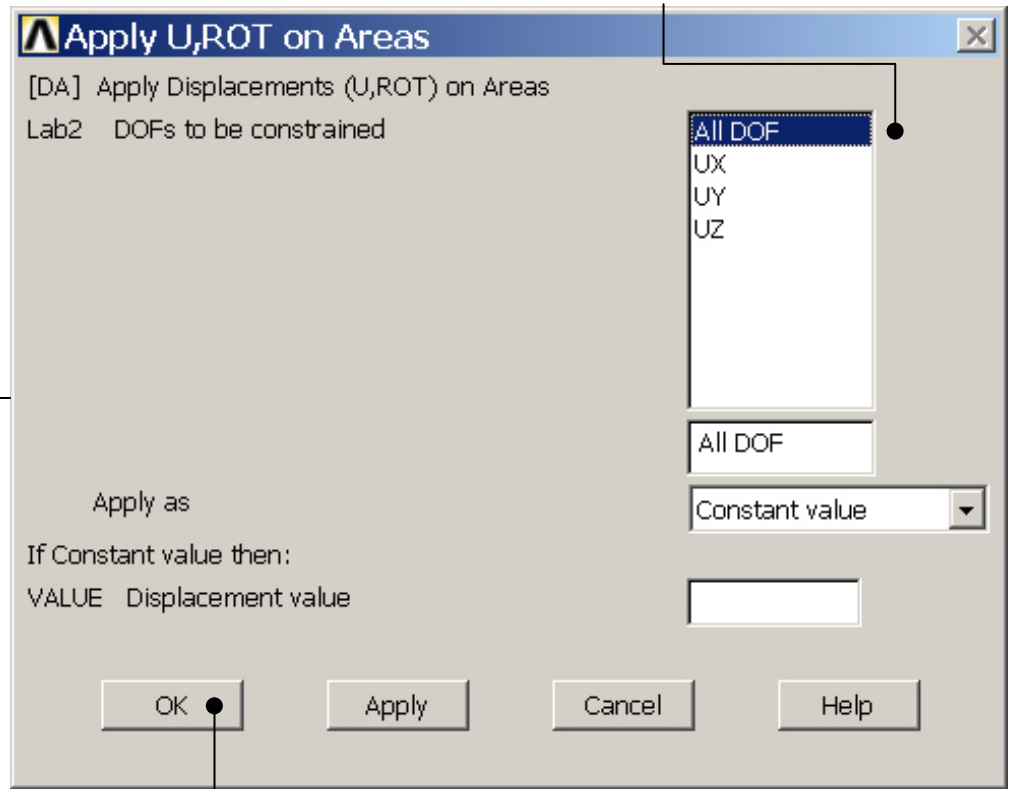
Example – Define Loads

Solution > Define Loads > Apply > Structural > Displacement > On Areas



Select Area A6 or the left end area

Select All DOF to fix/clamp the beam

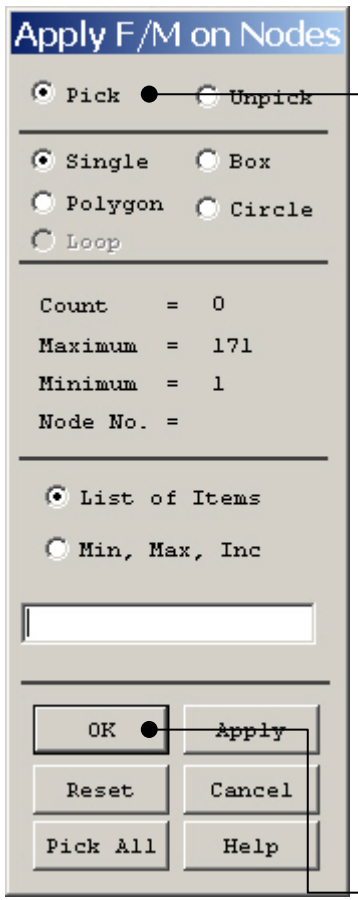


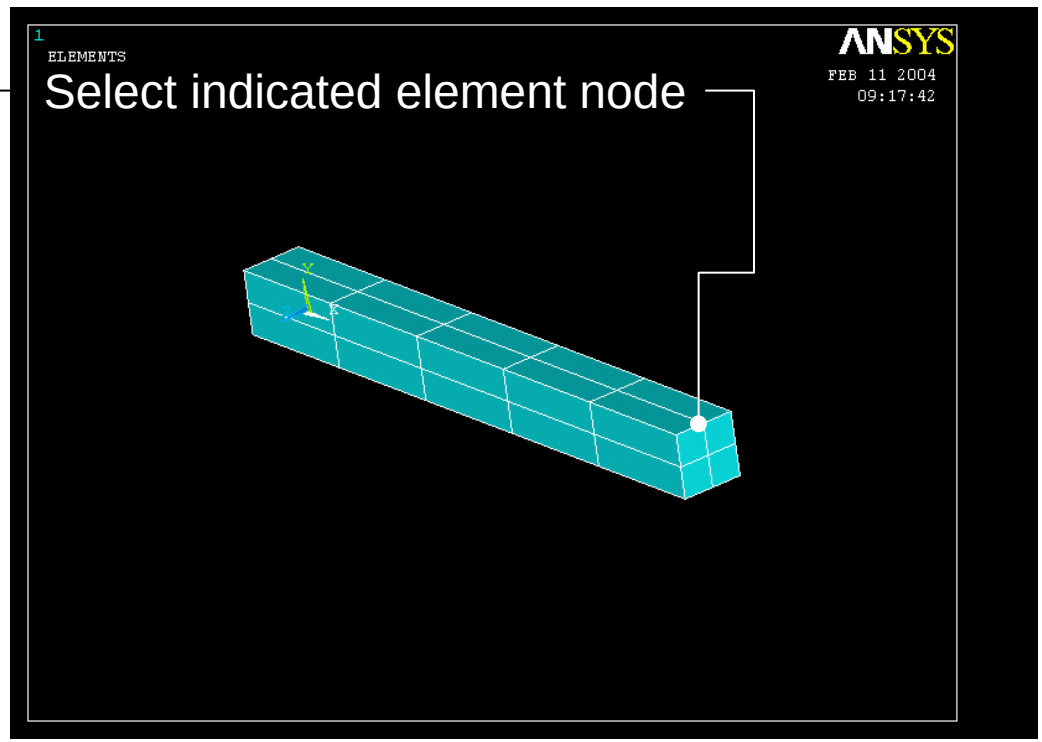
Press OK

Example – Define Loads

Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

Note: If the model is remeshed all loads will be deleted with the element nodes



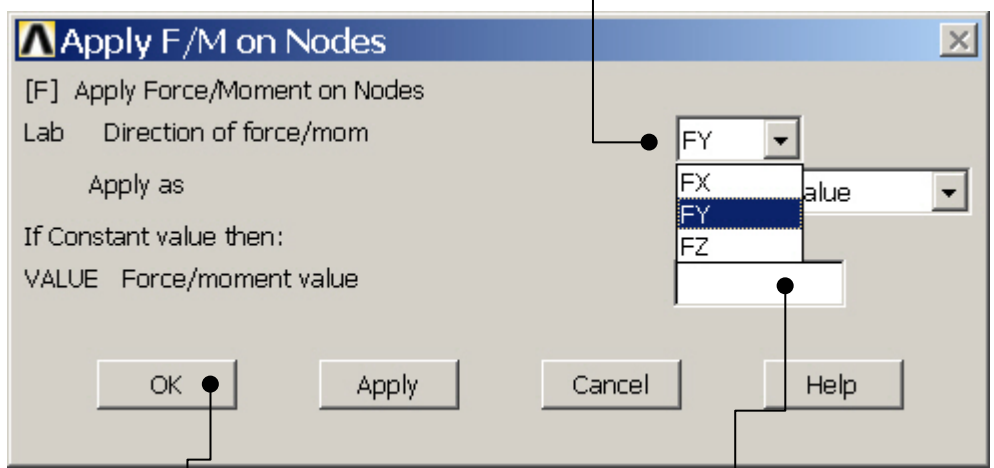


Press OK

Example – Define Loads

Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

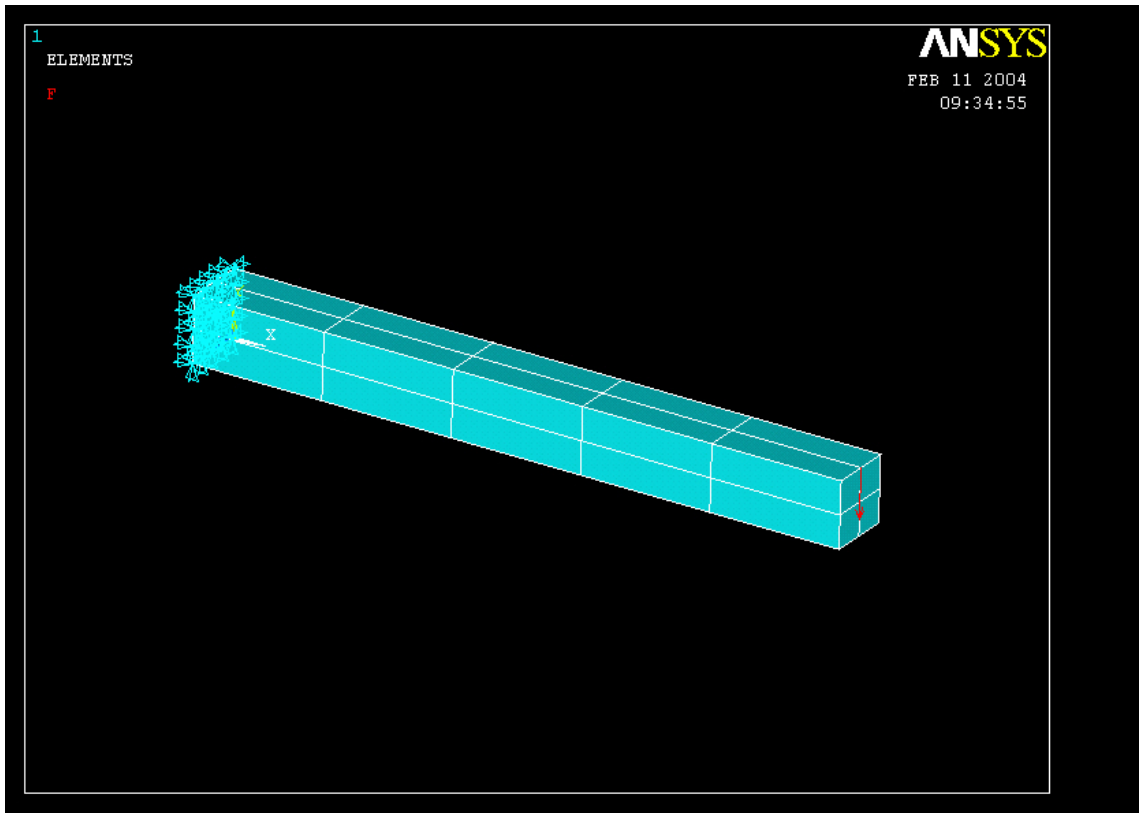
Change to FY



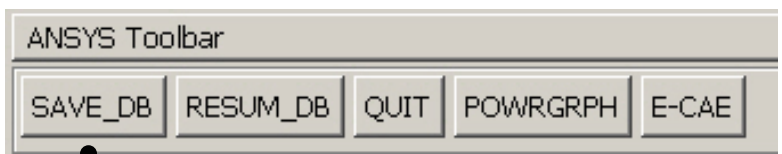
Press OK to finish

Enter -1000

Example - Save



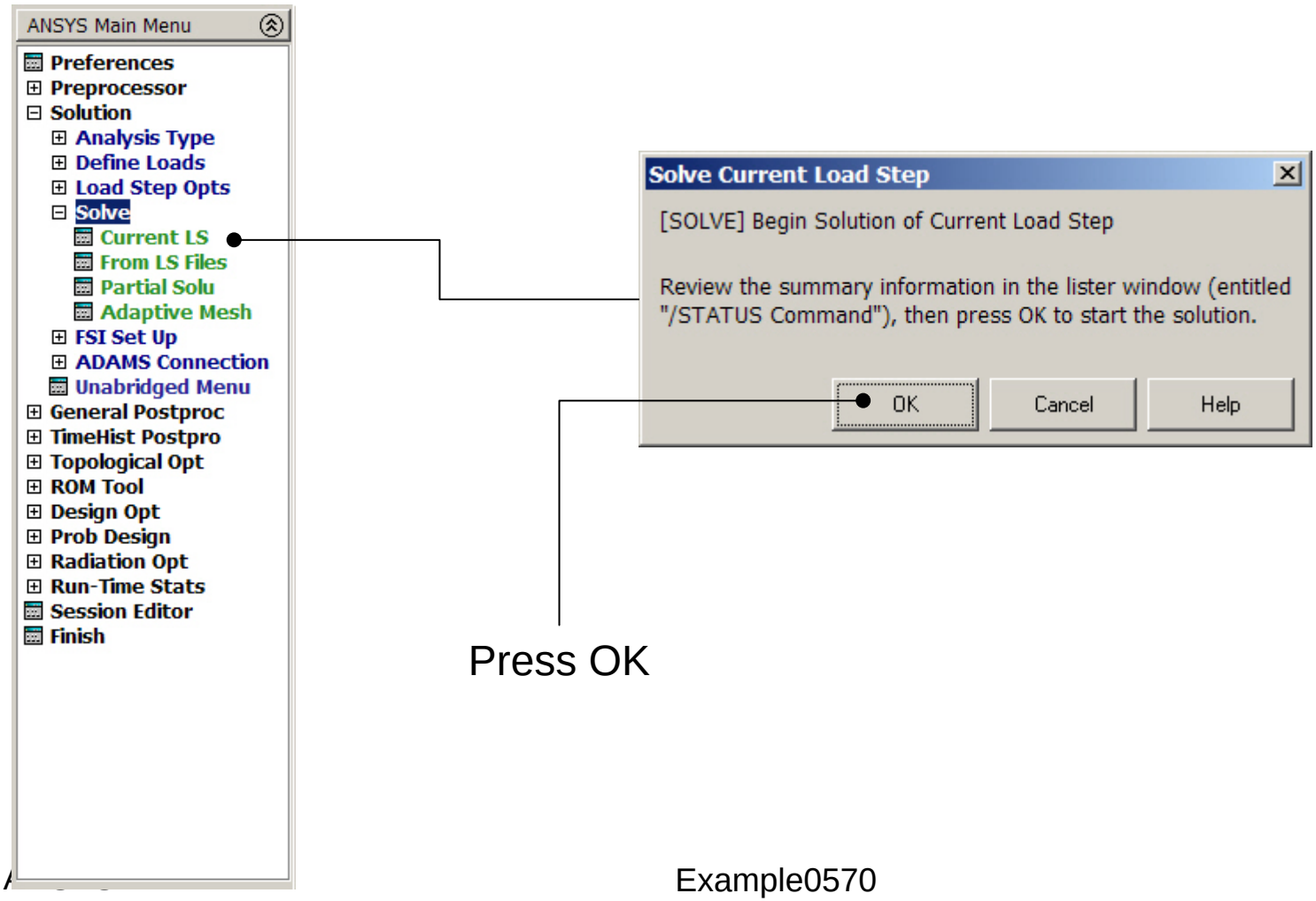
Display of Analysis model



Save the model

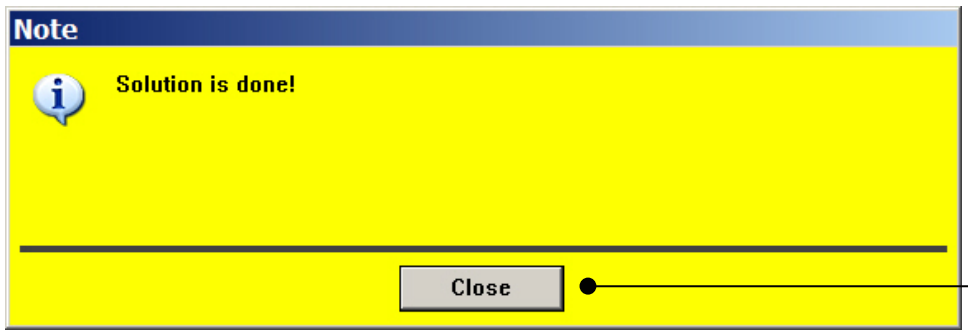
Example - Solve

Solution > Solve > Current LS



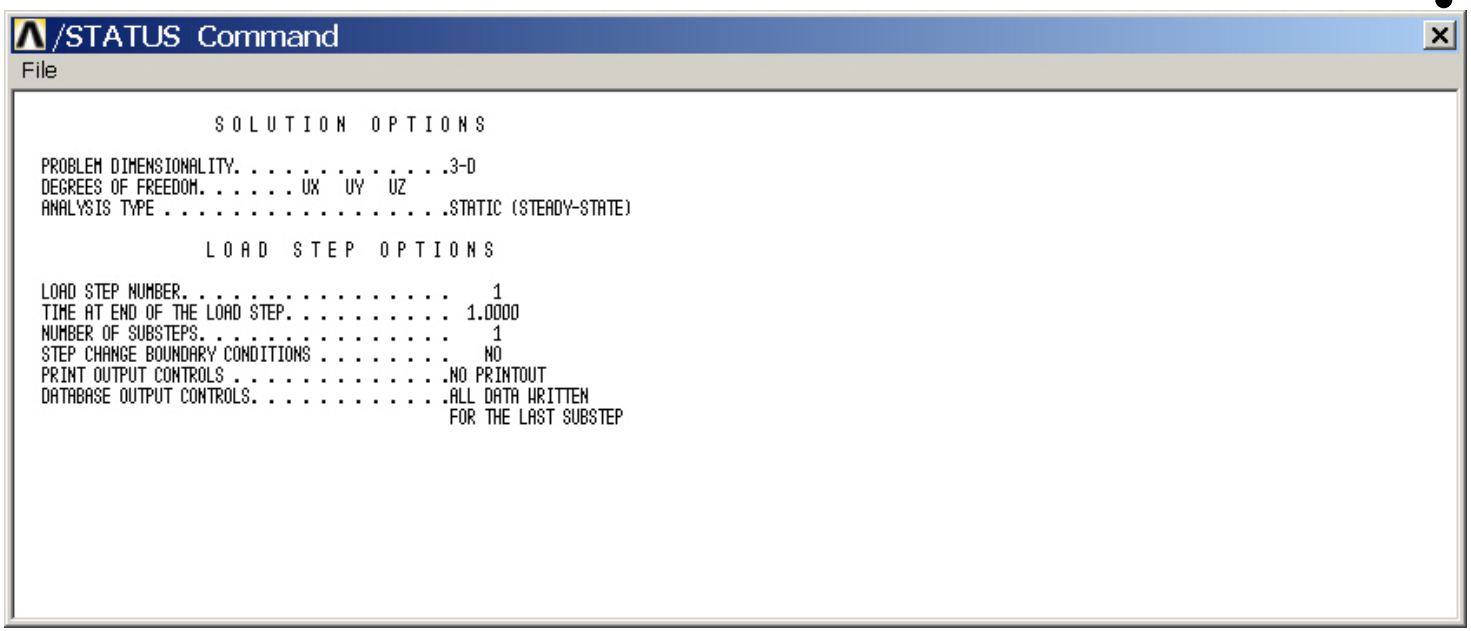
Press OK

Example - Solve



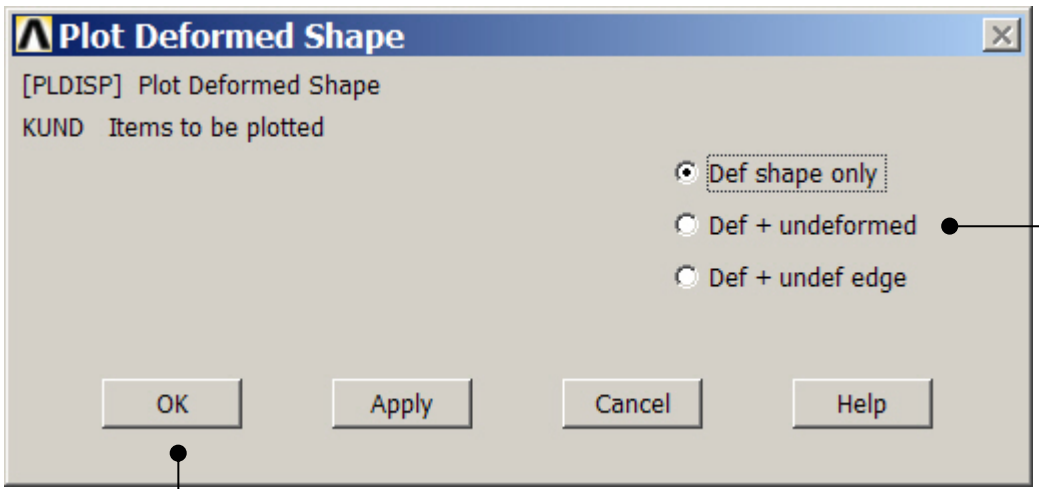
Press Close

Press here
to Close



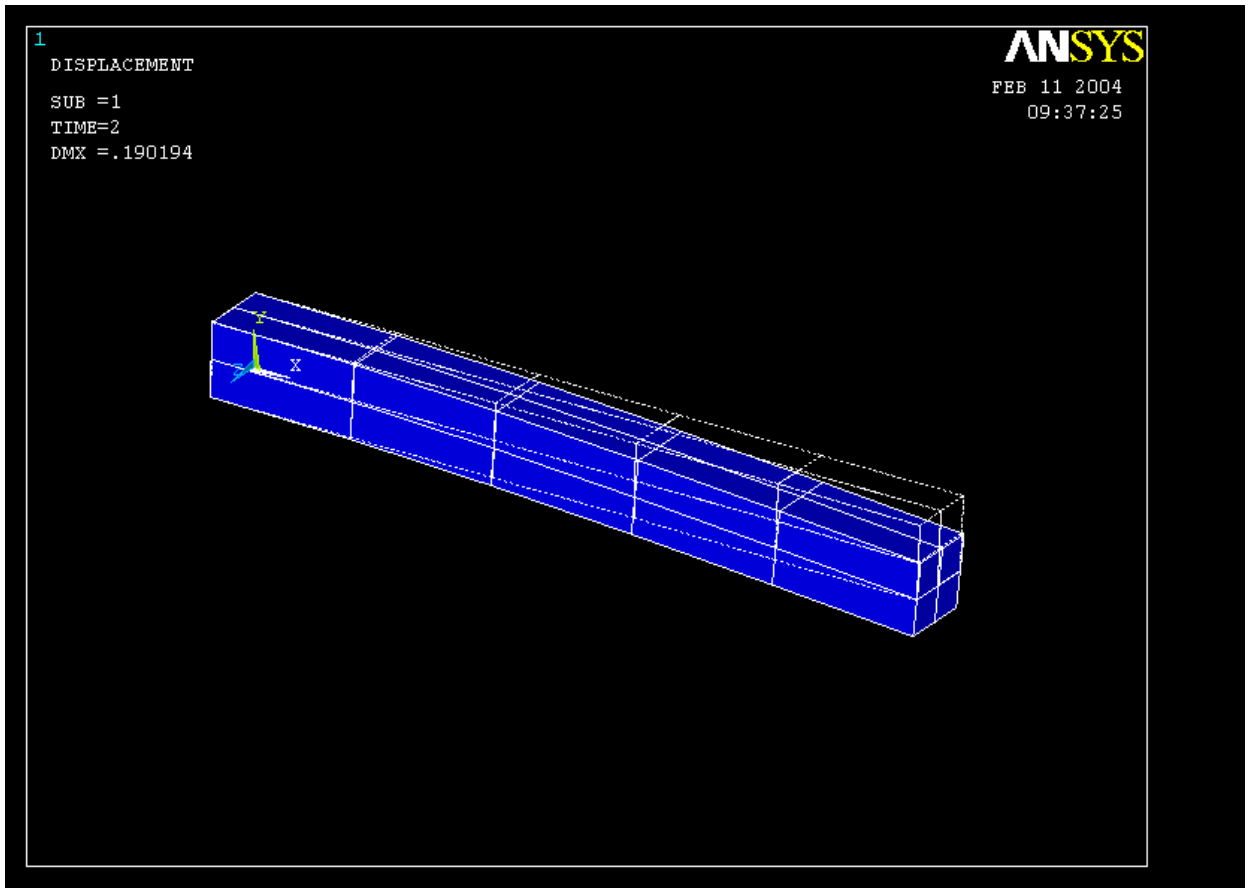
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



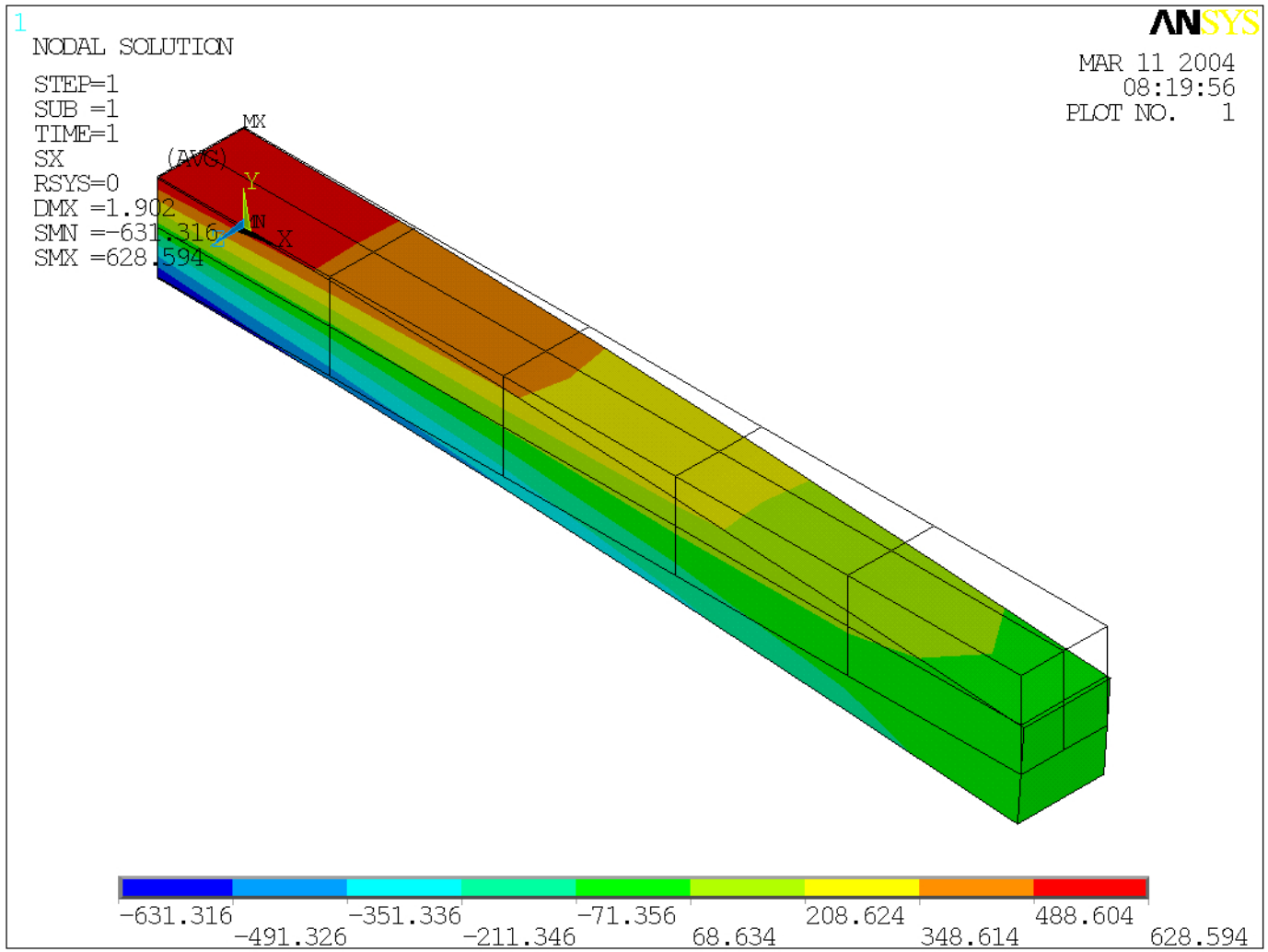
Select “Def+undeformed” and Press OK

Example - PostProcessing

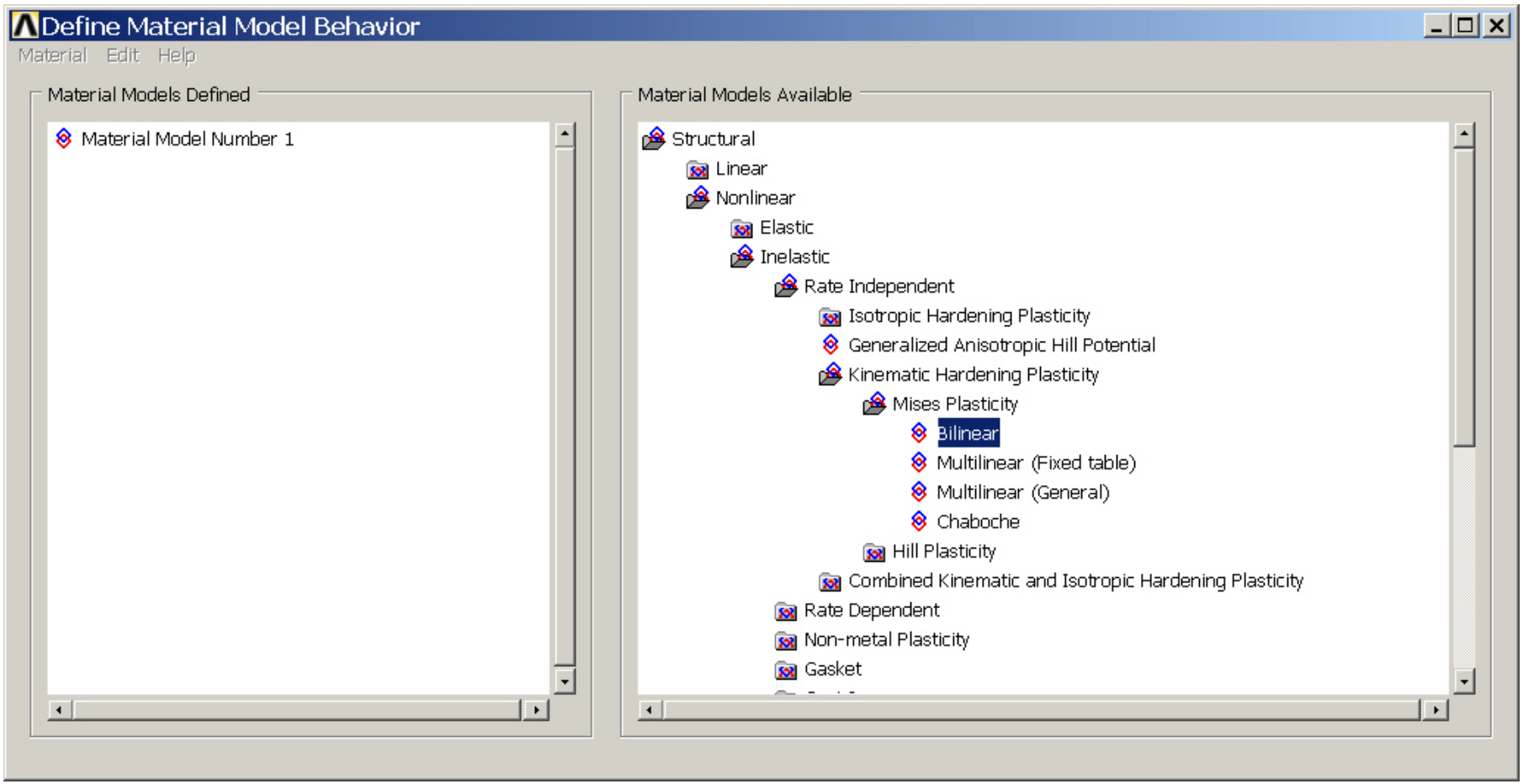


Read Maximum displacement: DMX

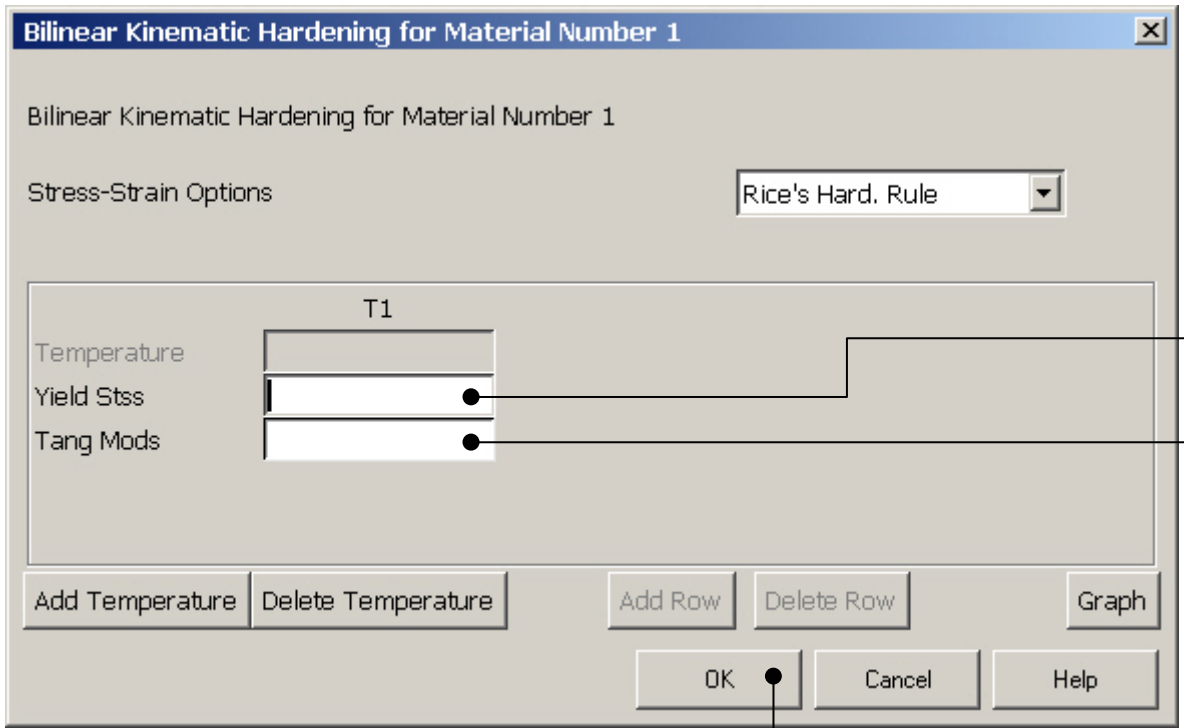
Example – Linear solution



Example – NL material models



Example – Bilinear kinematic hardening

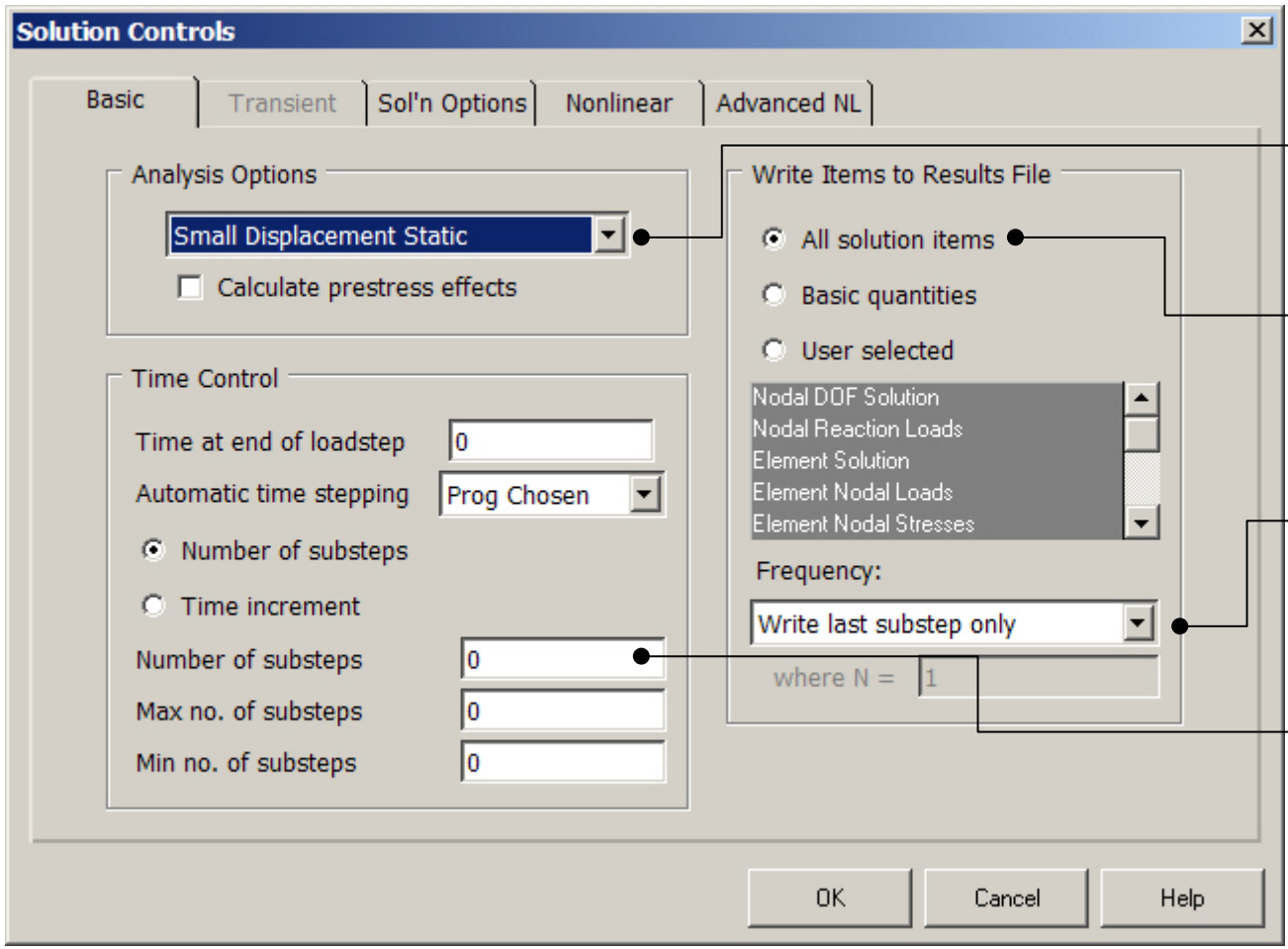


Enter 355 as the Yield Stress

Enter 1000 as the Tangent Modulus

Press OK

Example – Solution Controls



Change to Large Displacement Static

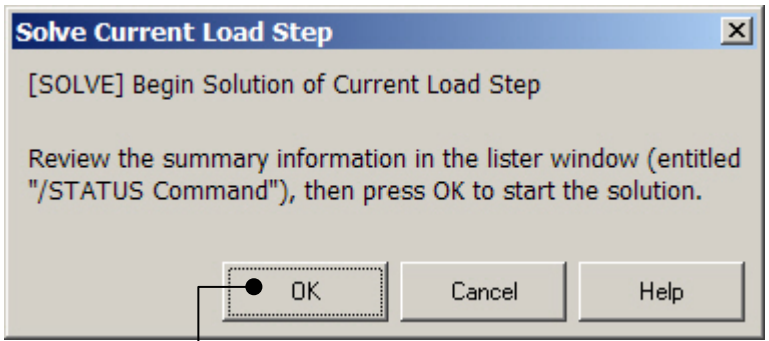
Select All solution items

Select Write every Nth substeps

Enter 30

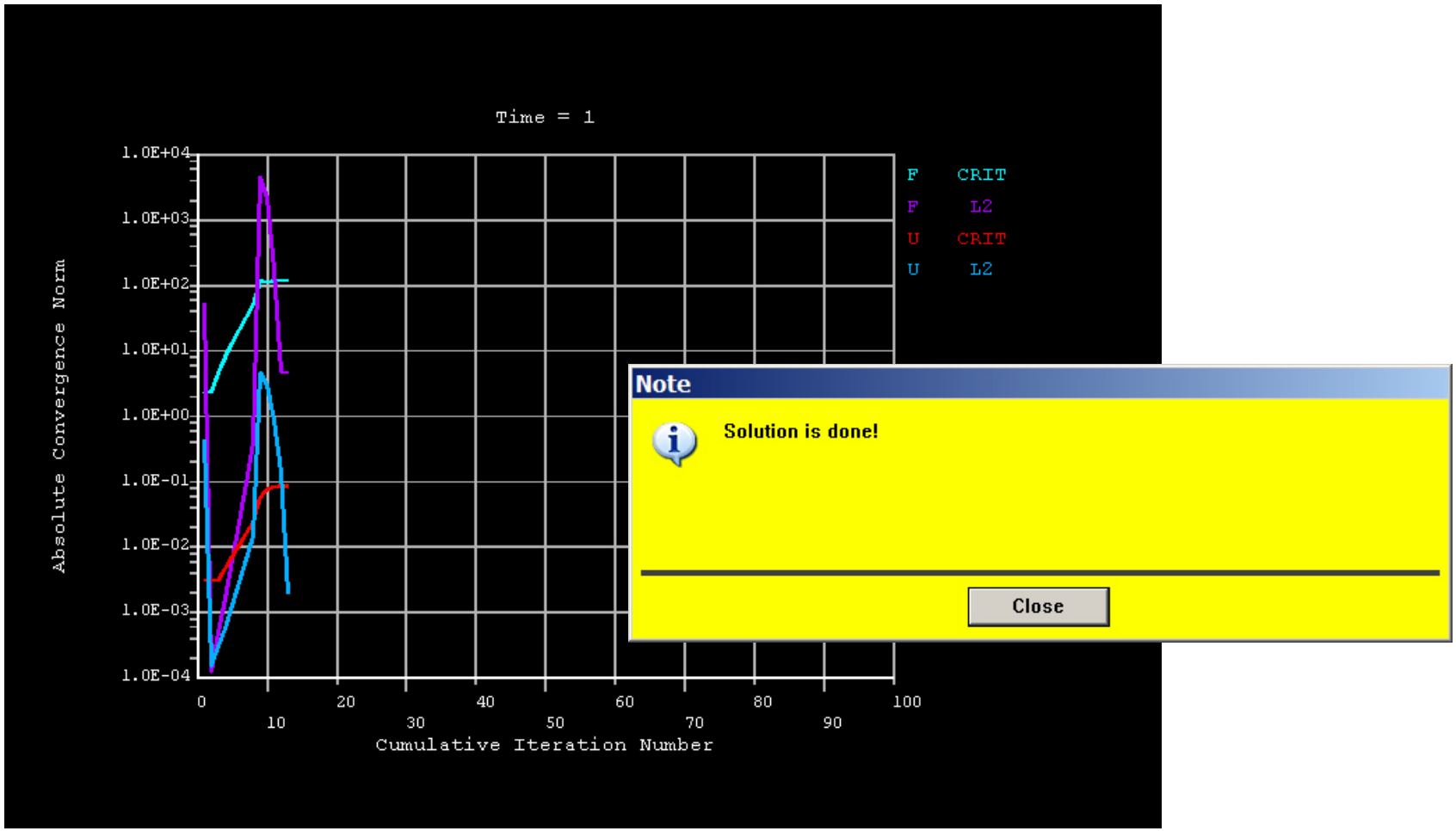
Example - Solve

Solution > Solve > Current LS



Press OK

Example - Convergence



Example – NL material solution