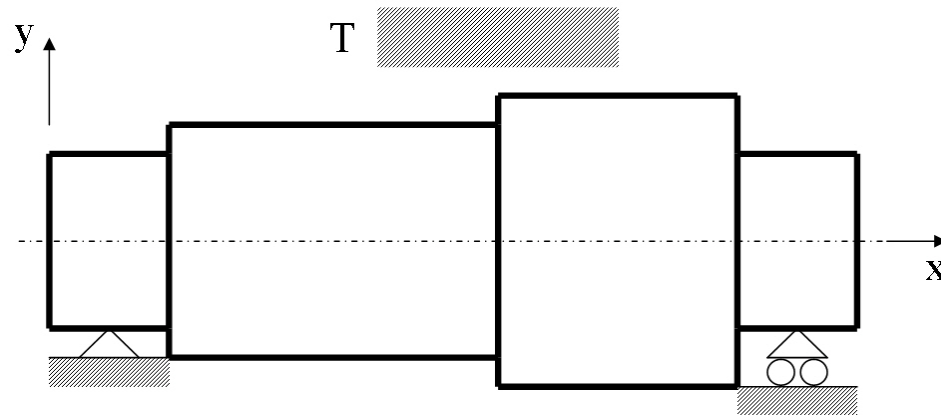


Course in ANSYS

Example0303

Example – Gear axle 3D

**Objective:**

Compute the maximum stress von Mises

Tasks:

How should this be modeled?

Topics:

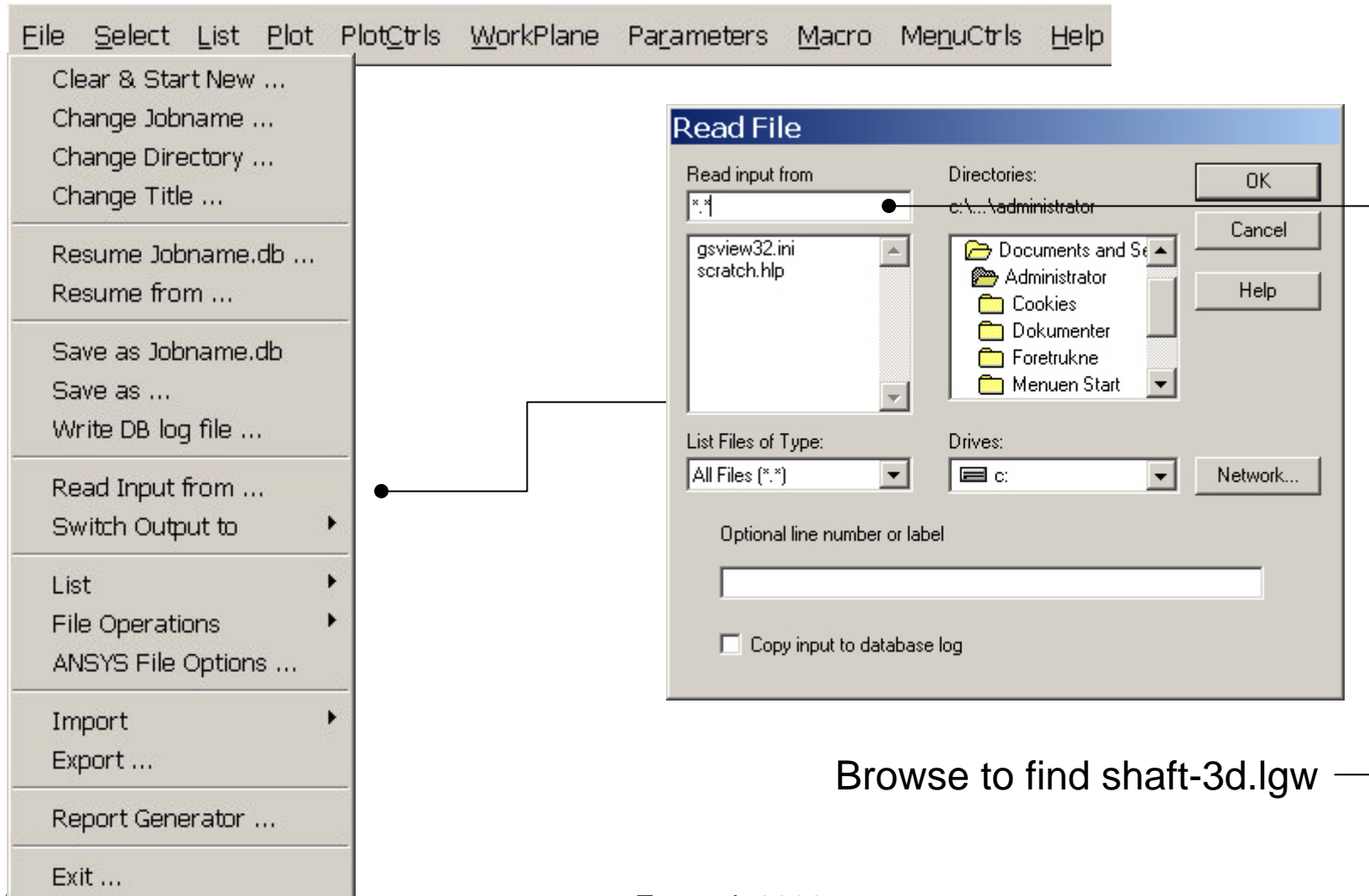
Element type, Real constants, modeling,
Plot results, output graphics, select entities

$$E = 210000 \text{ N/mm}^2$$

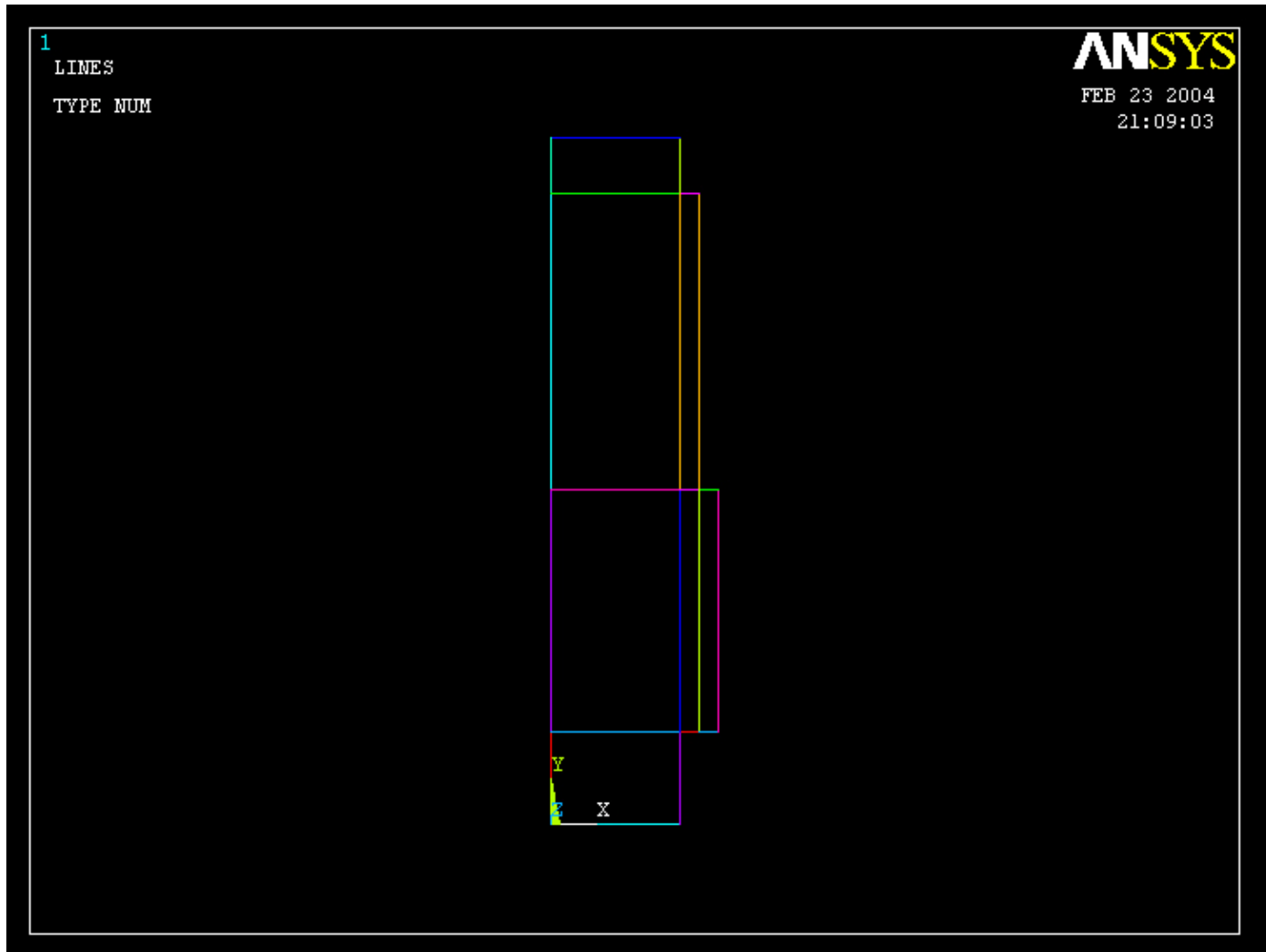
$$\nu = 0.3$$

$$T = 1000 \text{ Nmm}$$

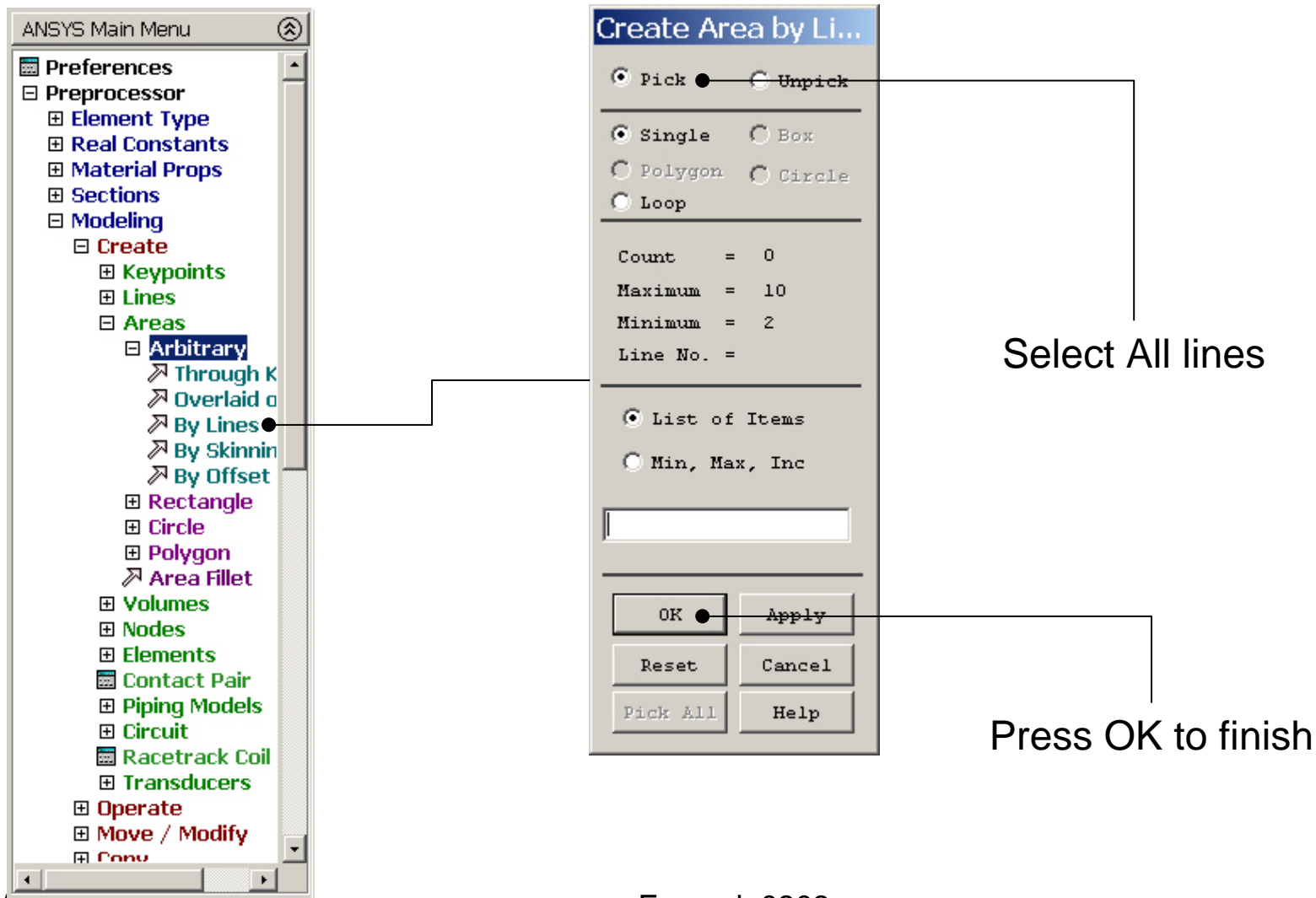
Example – Read Input from



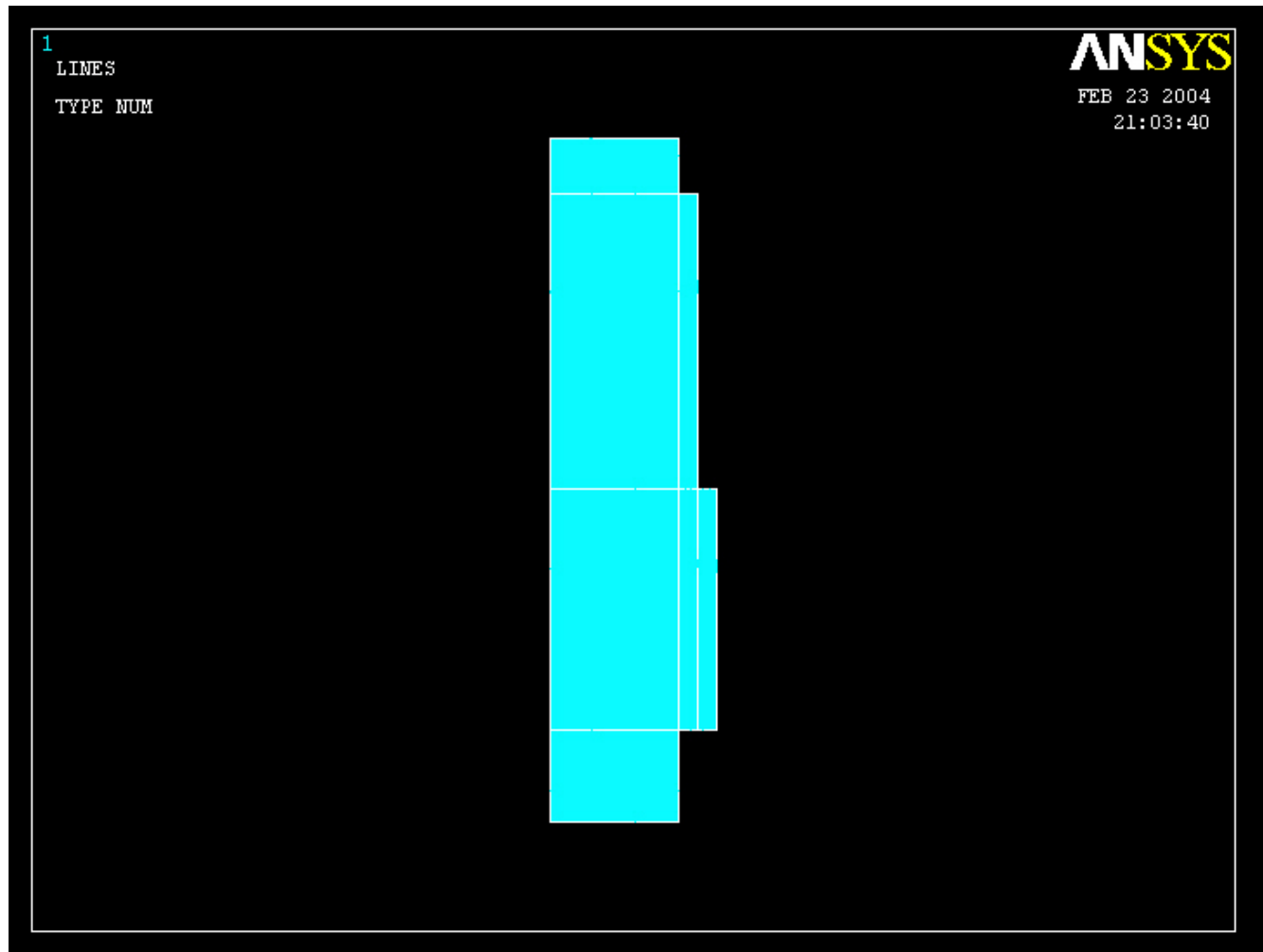
Example – Read Input from



Example – Create Areas by line

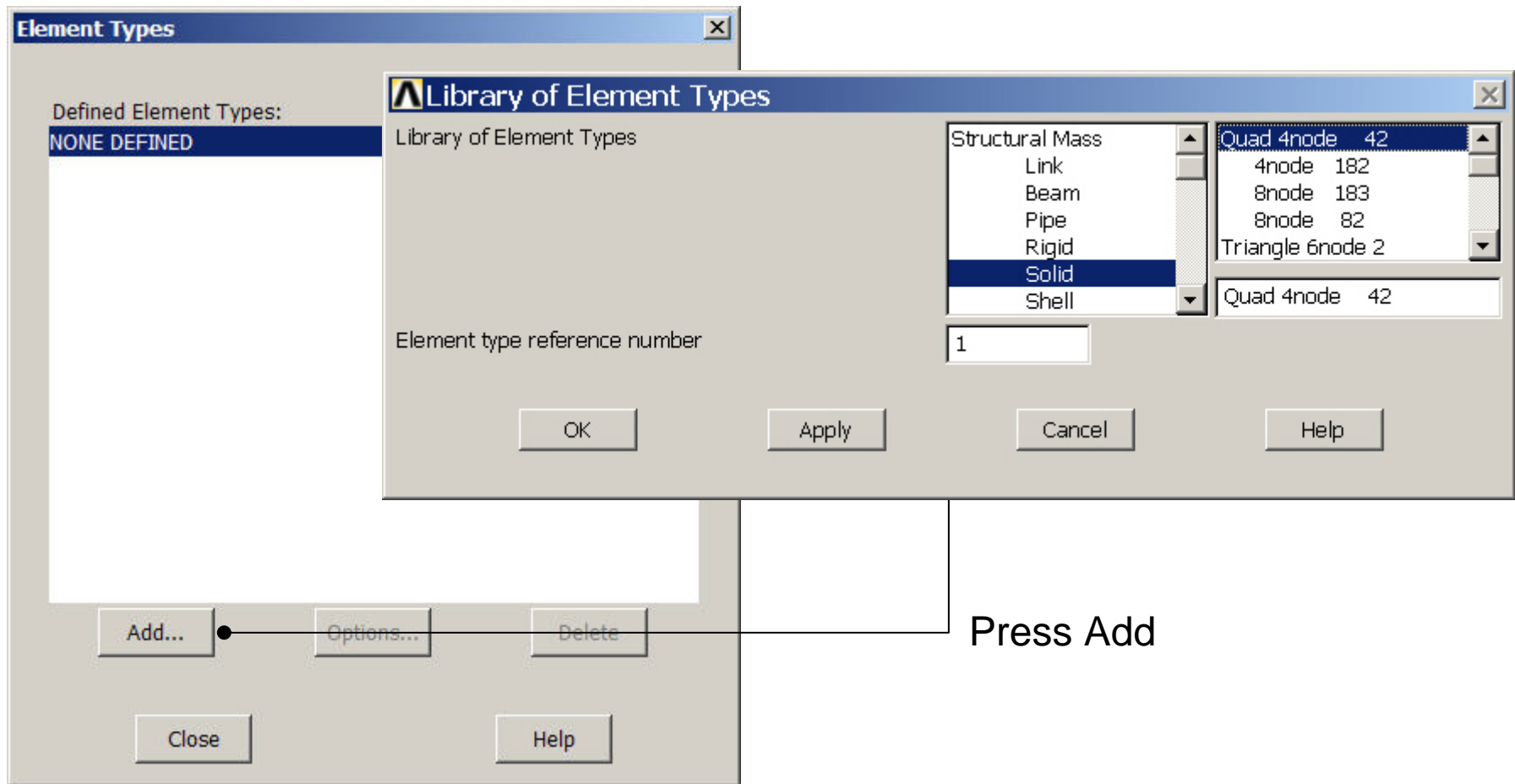


Example – Create Areas by line



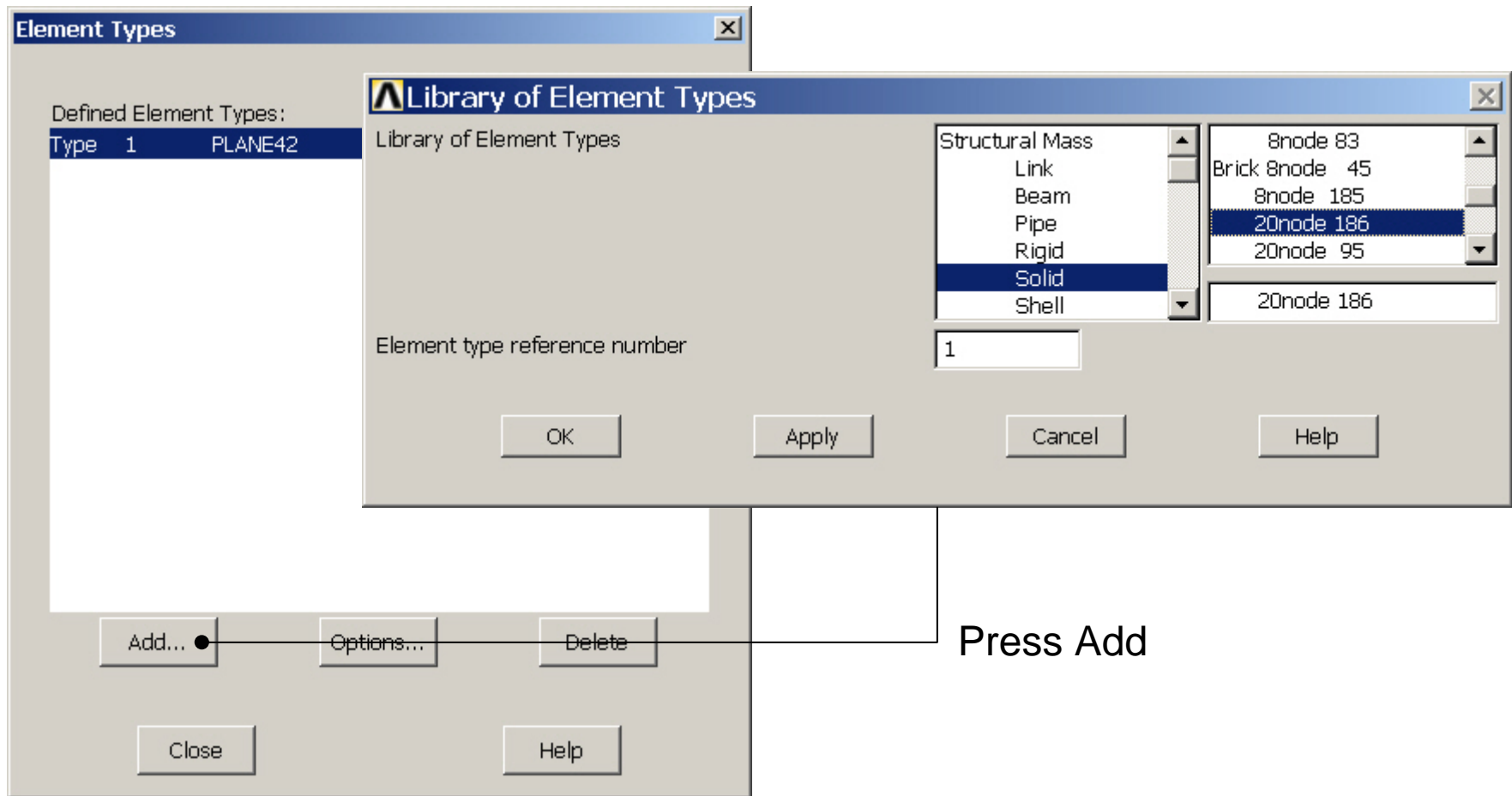
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete

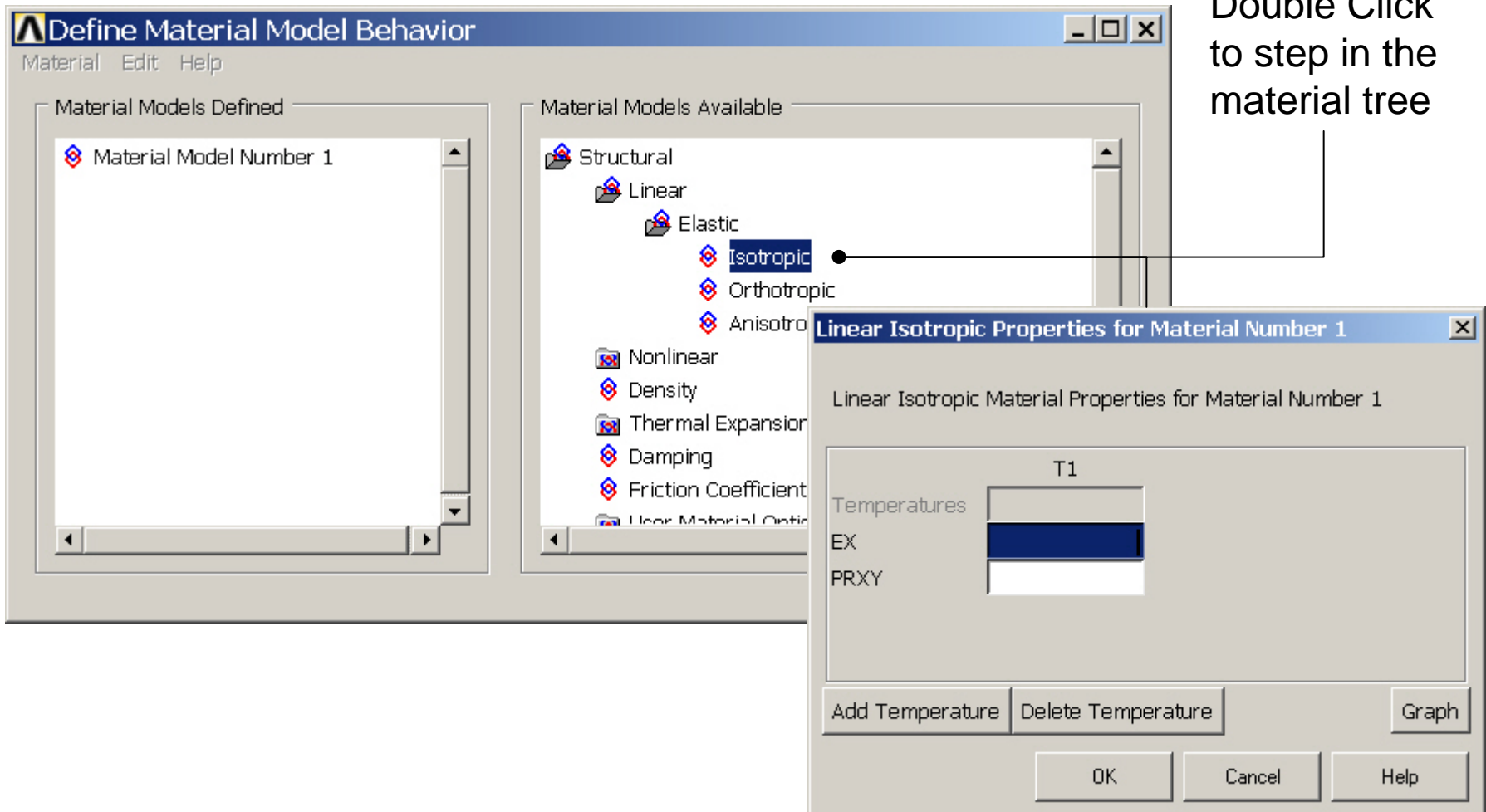


Example – Real Constants

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

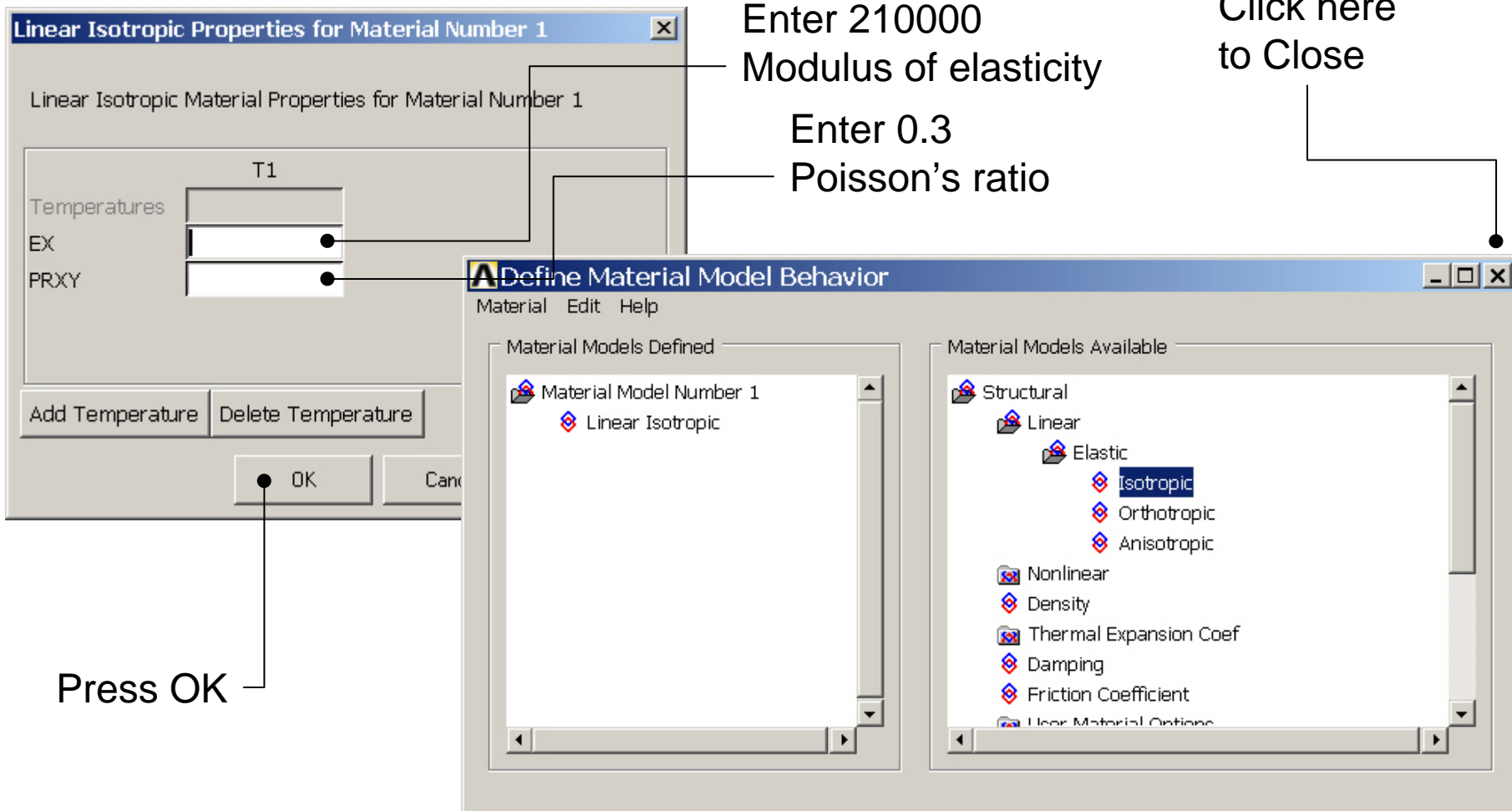
Example - Material Properties

Preprocessor > Material Props > Material Models



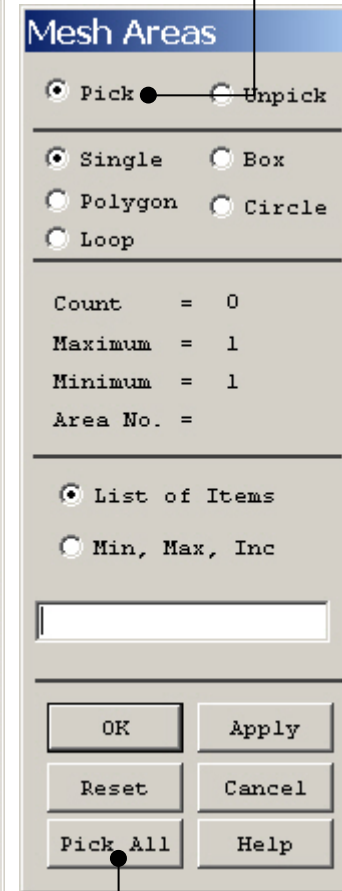
Example - Material Properties

Preprocessor > Material Props > Material Models



Example - Meshing

Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

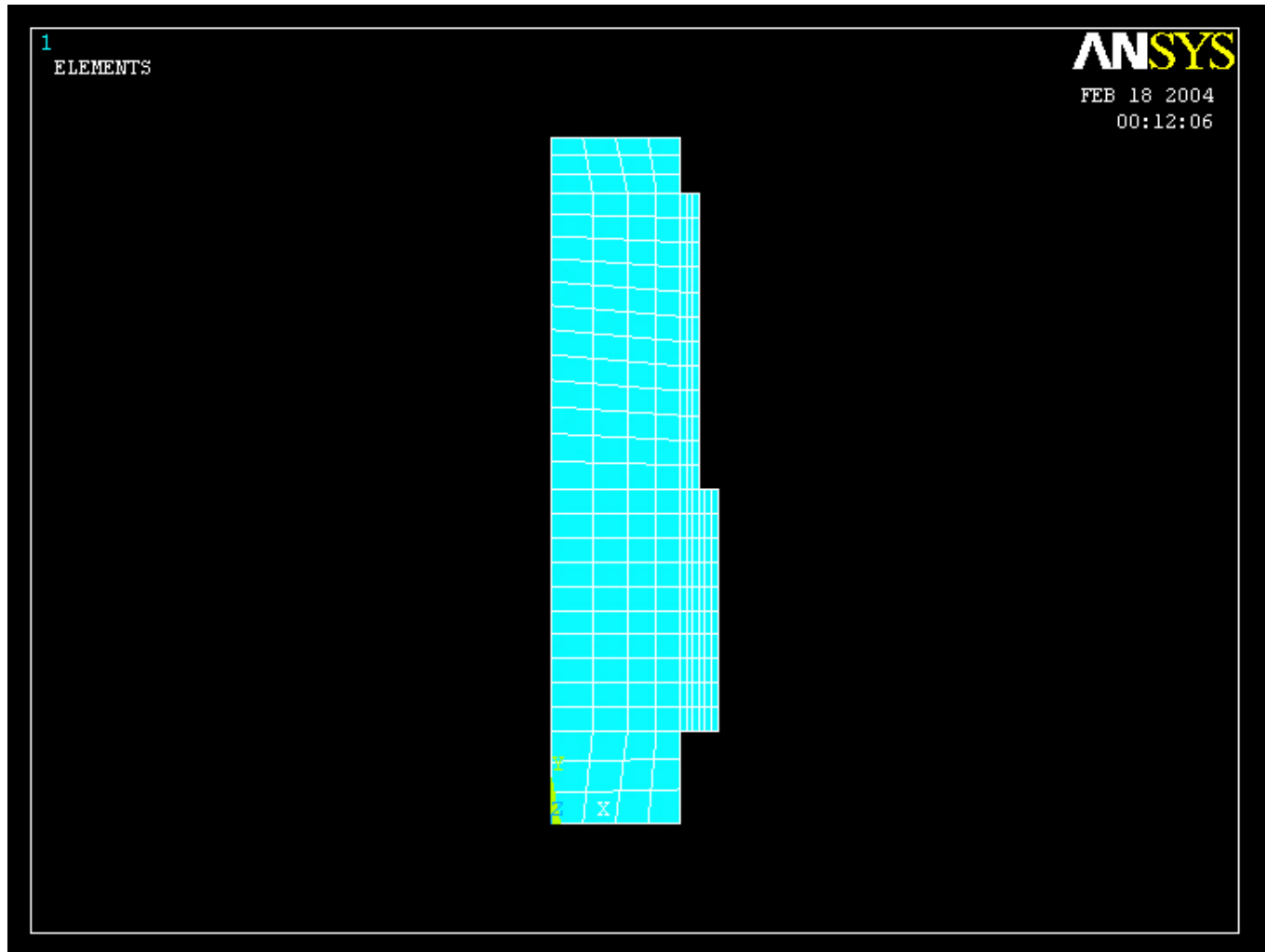


Select individual areas to be meshed

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

Select all areas defined to be meshed

Example - Meshing



Example – Extrude – About Axis

Select All Areas

Sweep Areas abo...

☒ Pick ☐ Unpick

☒ Single ☐ Box

☐ Polygon ☐ Circle

☐ Loop

Count = 0
Maximum = 1
Minimum = 1
Area No. =

☒ List of Items

☐ Min, Max, Inc

OK Apply

Reset Cancel

Pick All Help

Sweep Areas abo...

☒ Pick ☐ Unpick

☐ Single ☐ Box

☐ Polygon ☐ Circle

☐ Loop

Count = 0

M [VROTAT] Sweep Areas about Axis

ARC Arc length in degrees 360

NSEG No. of volume segments

OK Apply Cancel Help

OK Apply

Reset Cancel

Pick All Help

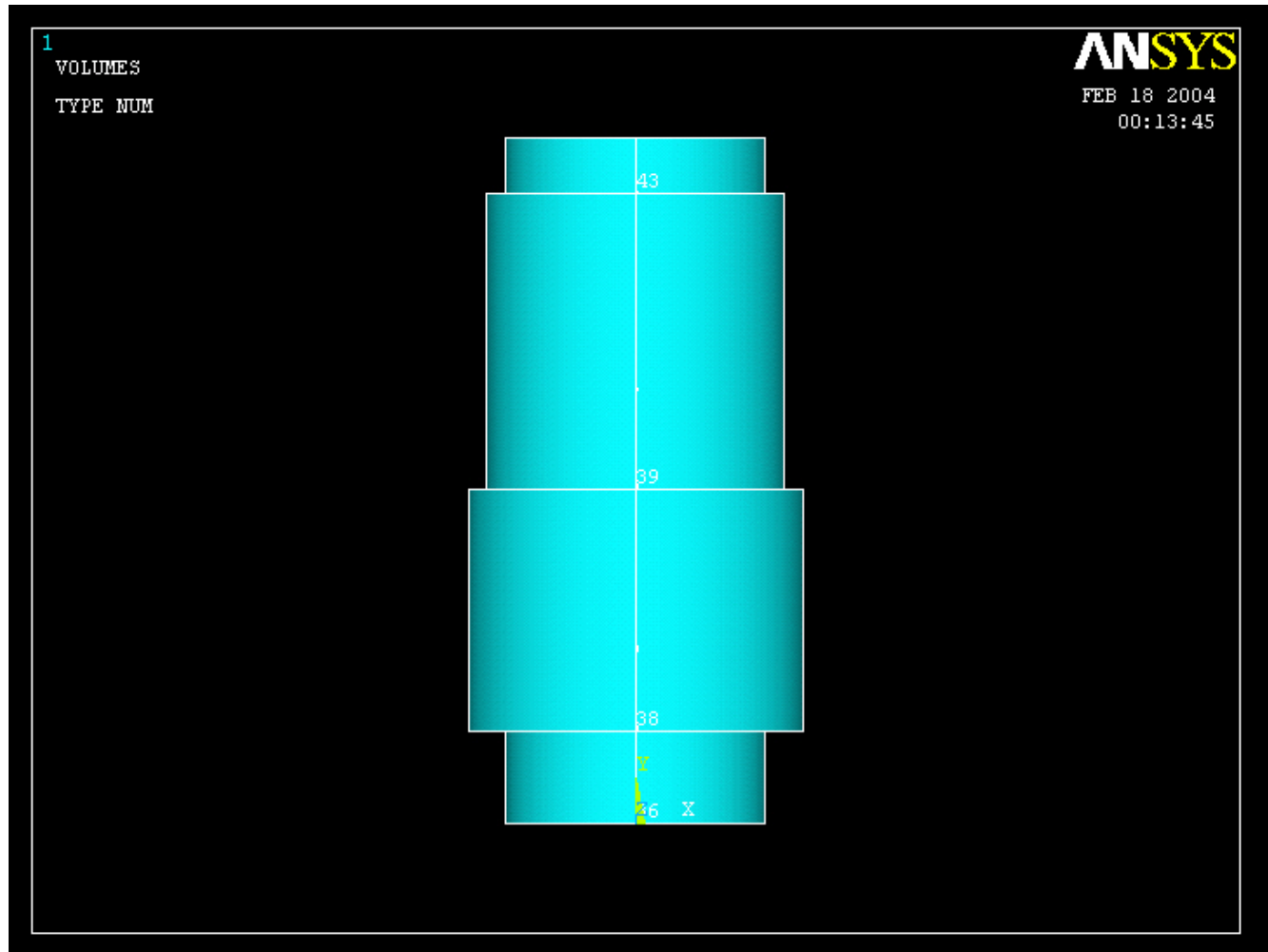
Pick or enter two keypoints defining the axis

Press OK

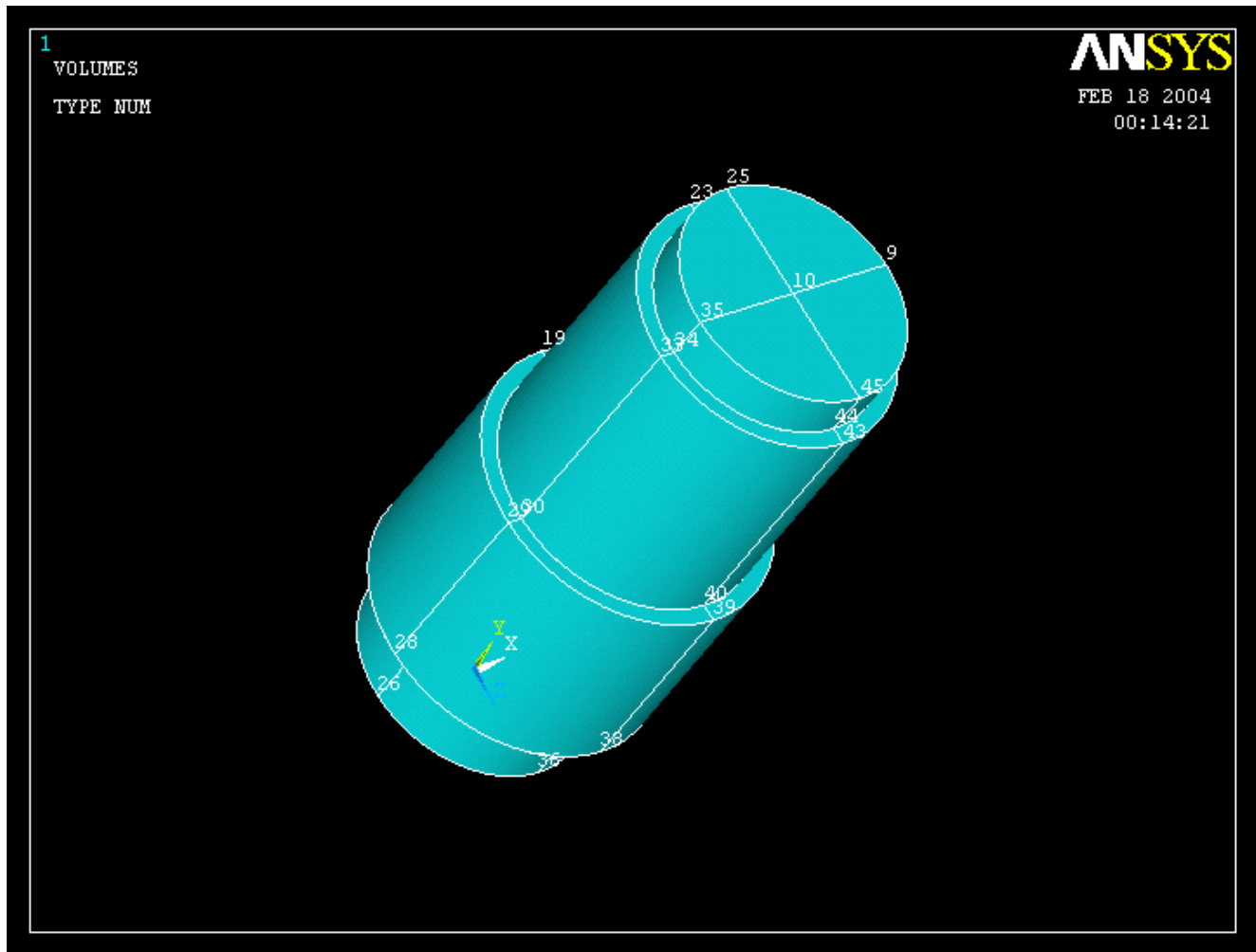
Note: Bottom left corner of ANSYS GUI

[VROTAT] Pick or enter areas to be swept about axis

Example – Extrude – About Axis

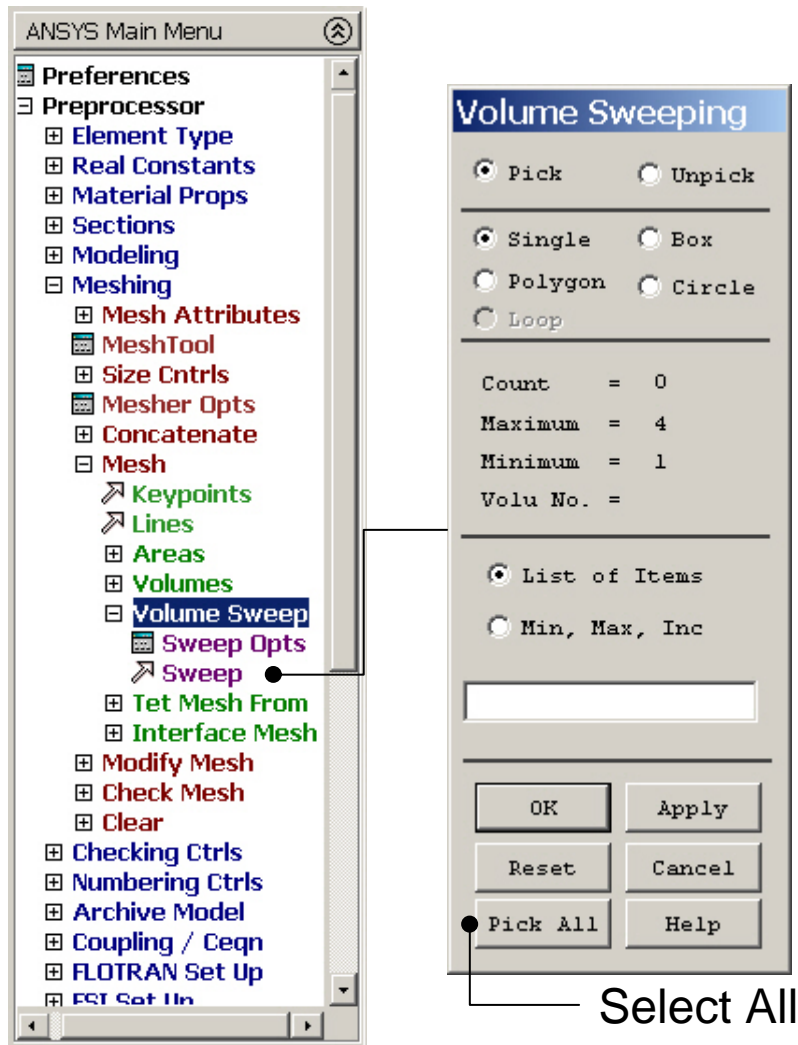


Example – Mouse rotate

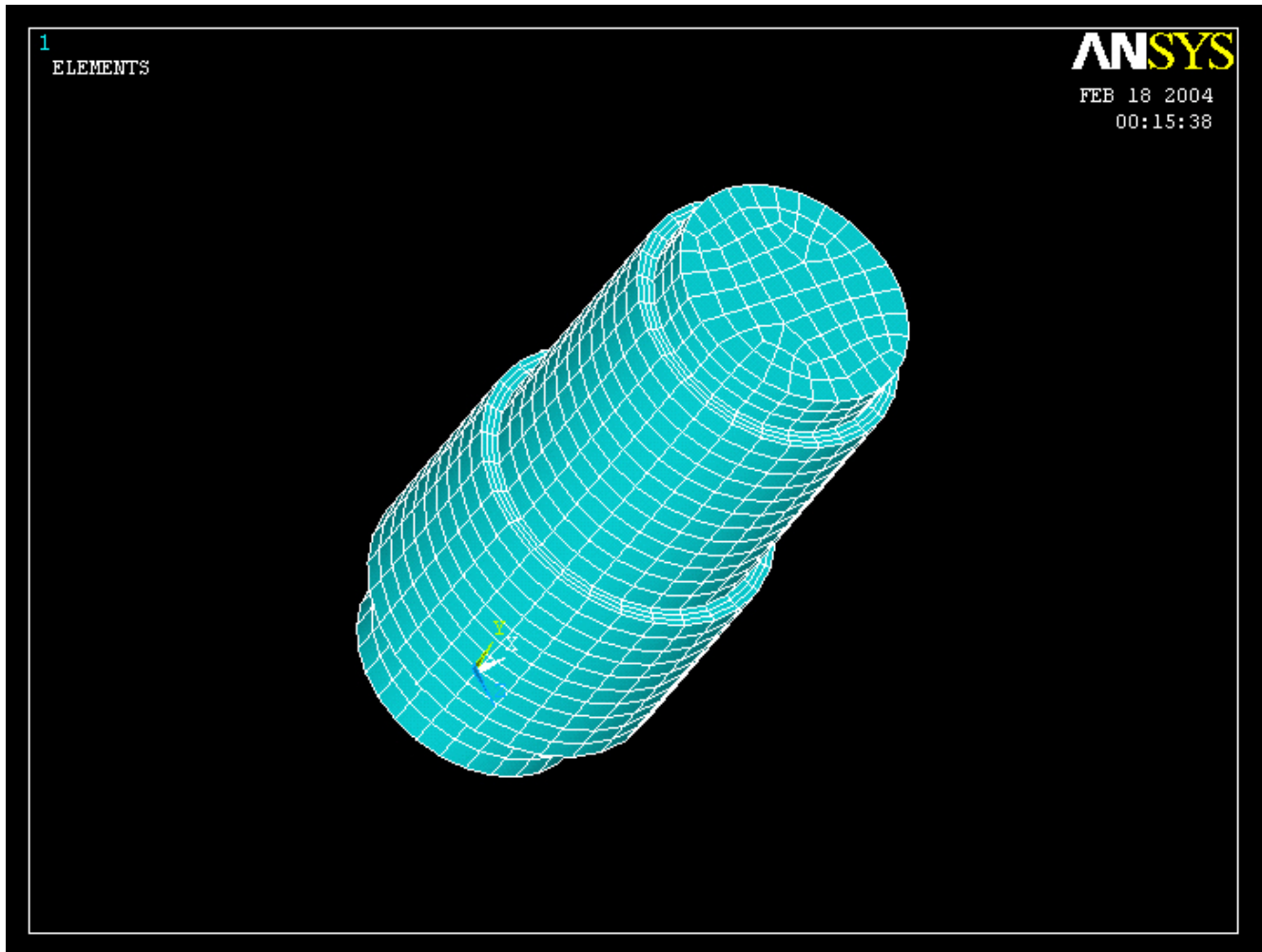


Rotate by
holding the
Ctrl key
down while
using the
right hand
mouse
button

Example – Volume Sweep

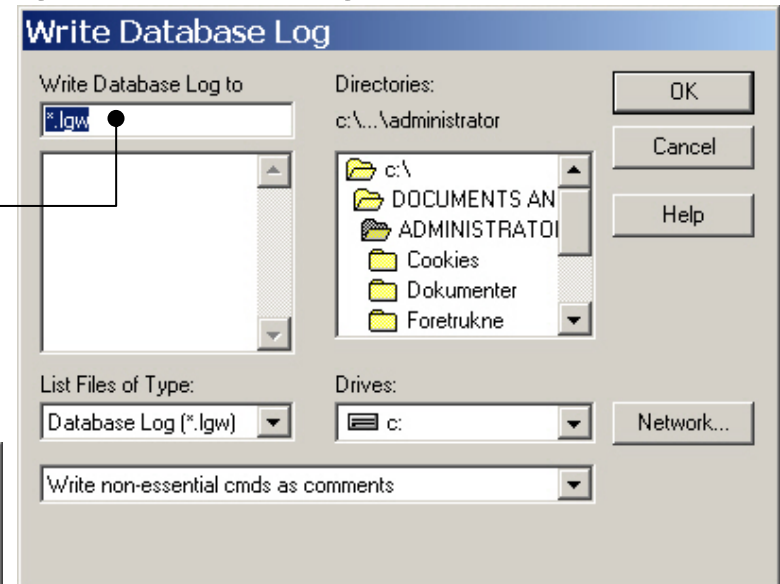


Example – 3D Mesh

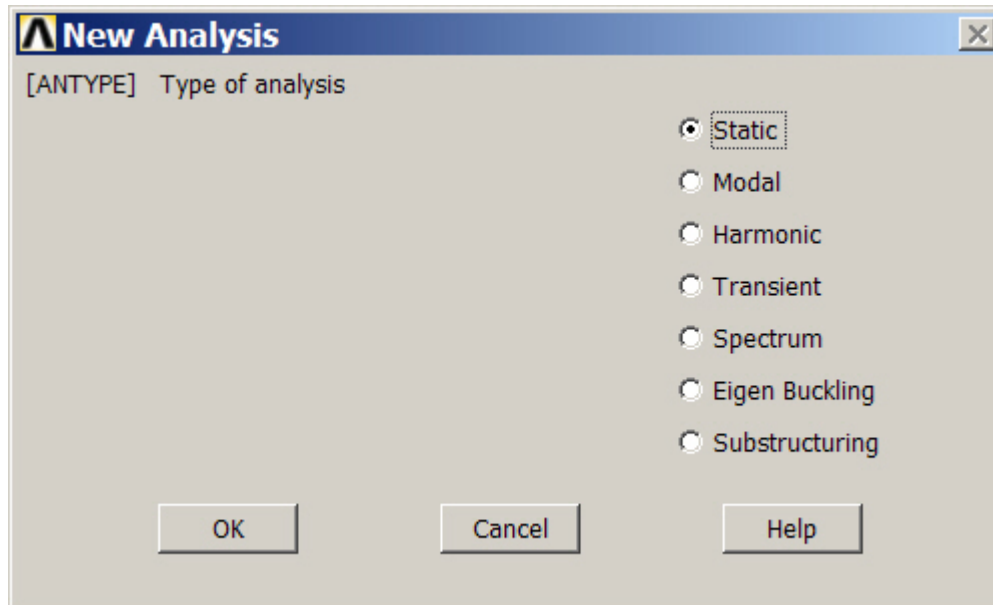


Example – Analysis Type

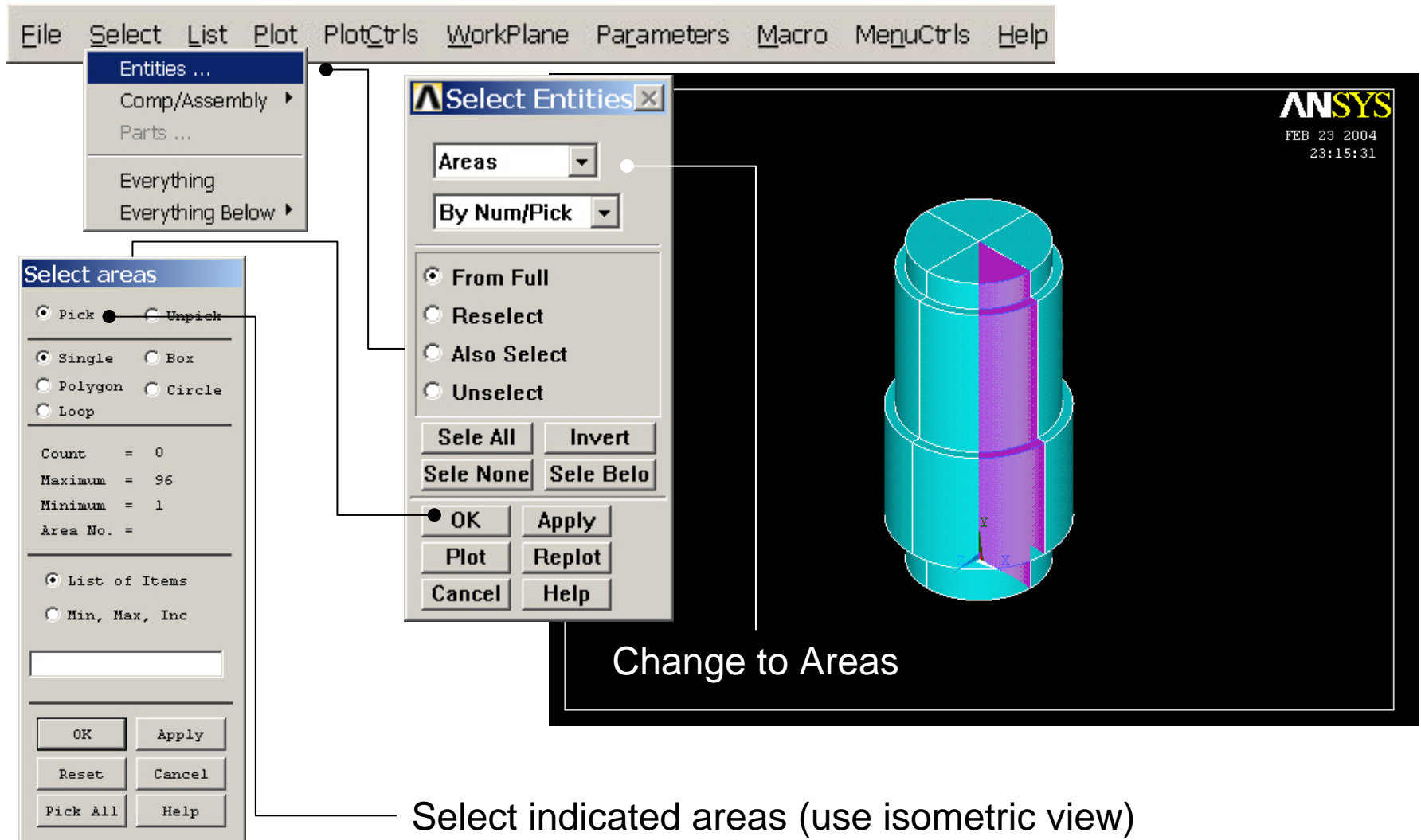
File > Write DB log file
Enter “example0303.lgw”



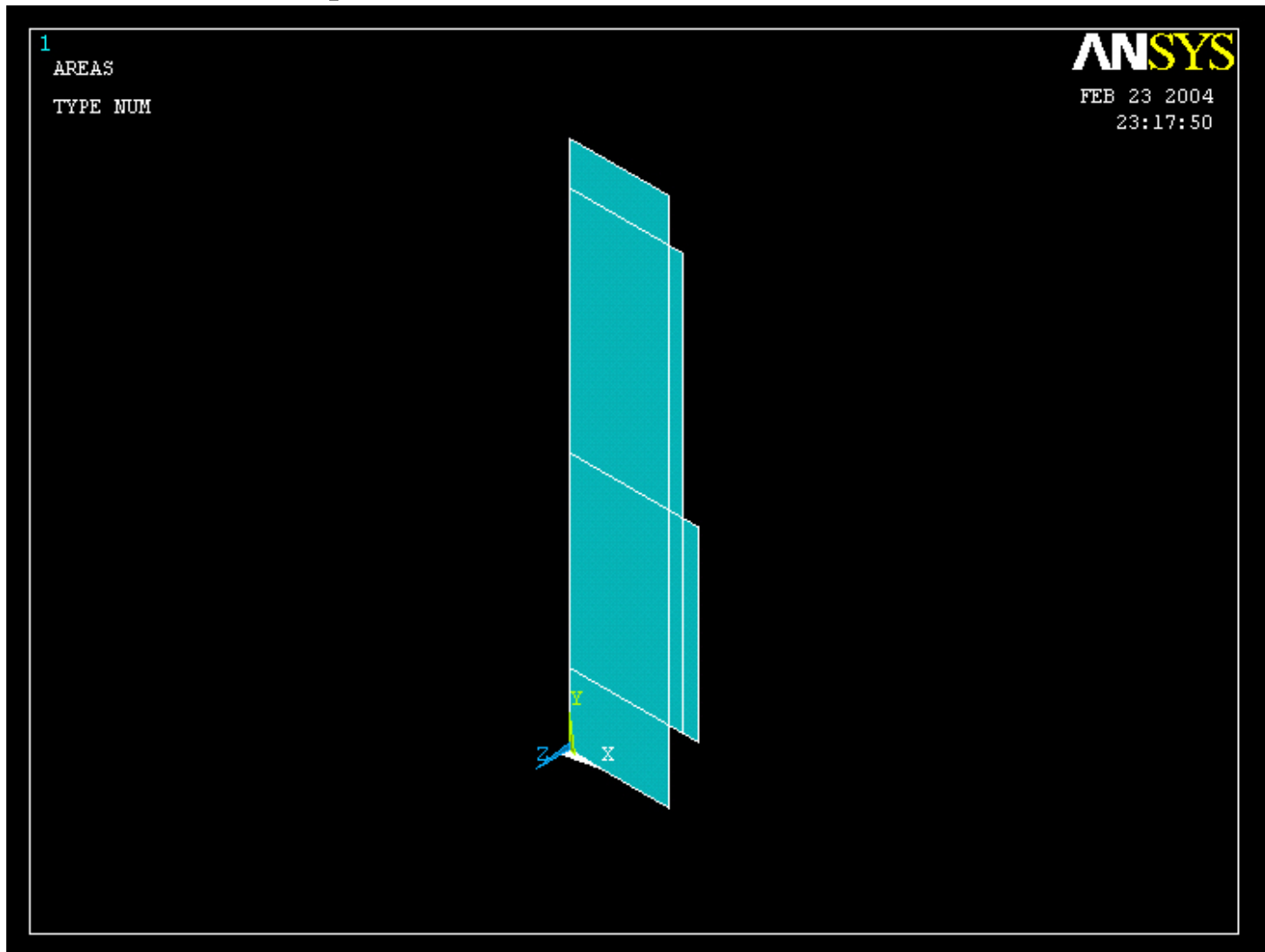
Solution > Analysis Type > New Analysis



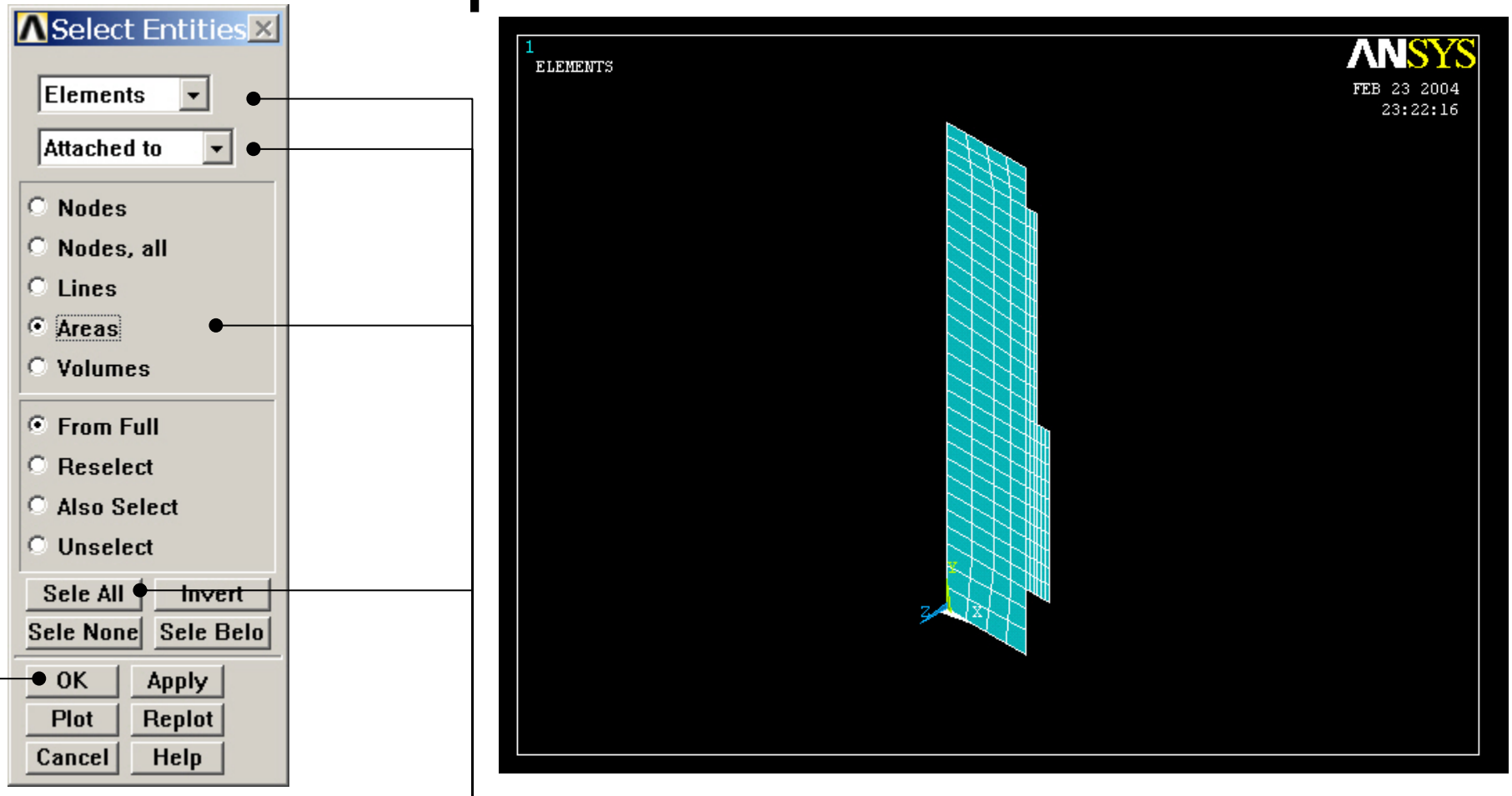
Example – Select entities



Example – Select entities



Example – Select entities



Change to Elements, Attached to, Areas, Sele All

Press OK

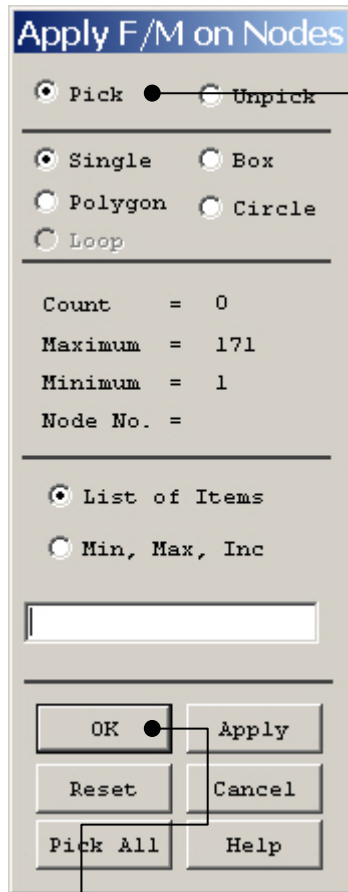
Computational Mechanics, AAU, Esbjerg

Example0303

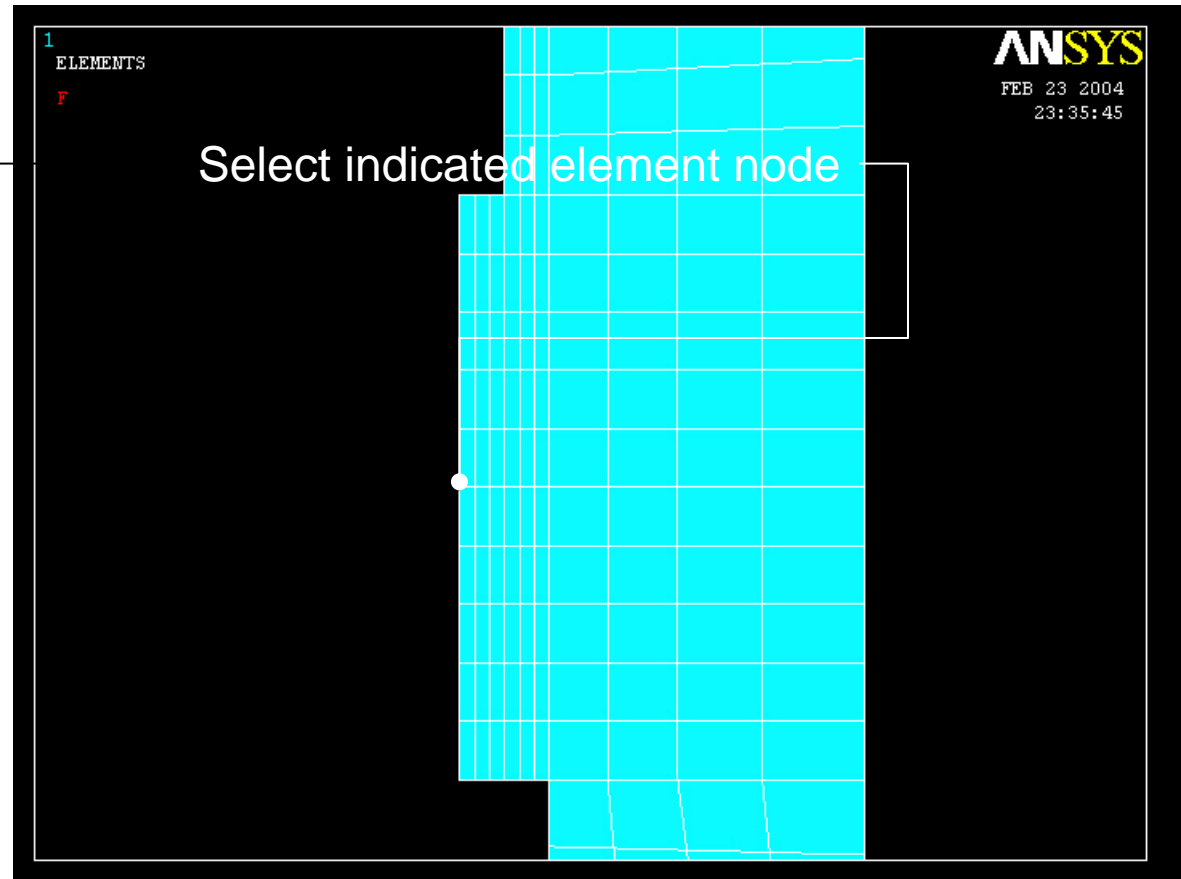
Example – Define Loads

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

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

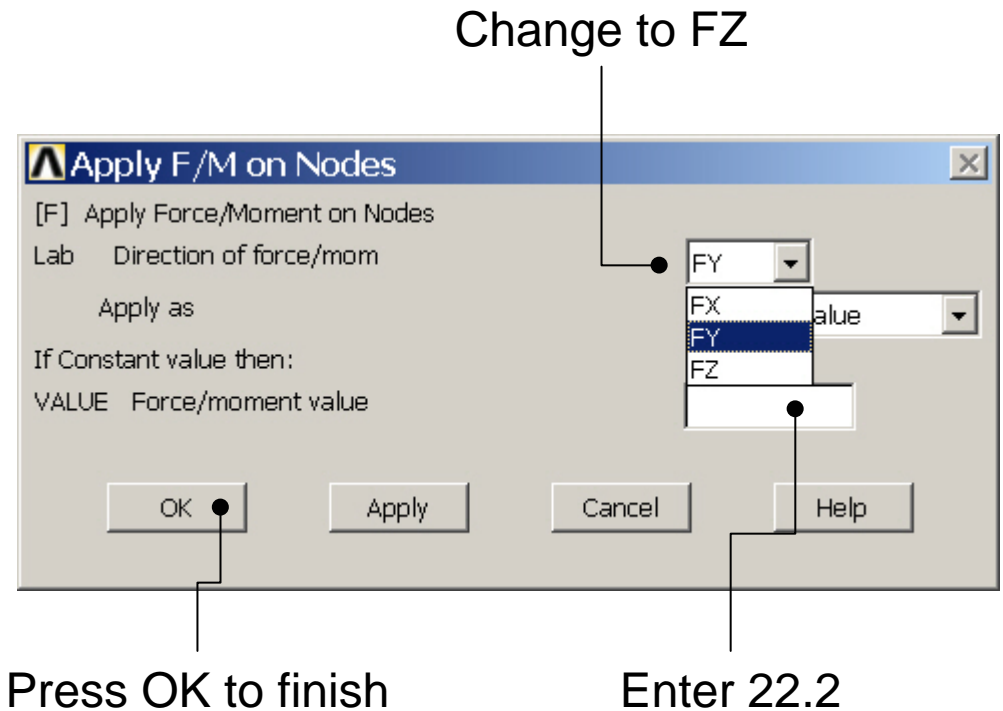


Press OK



Example – Define Loads

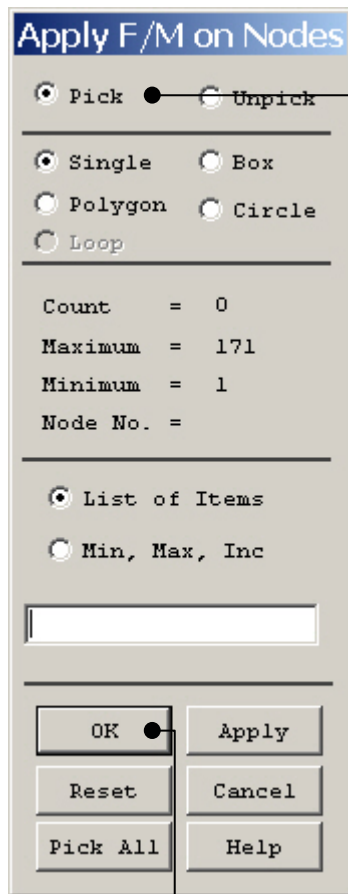
Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes



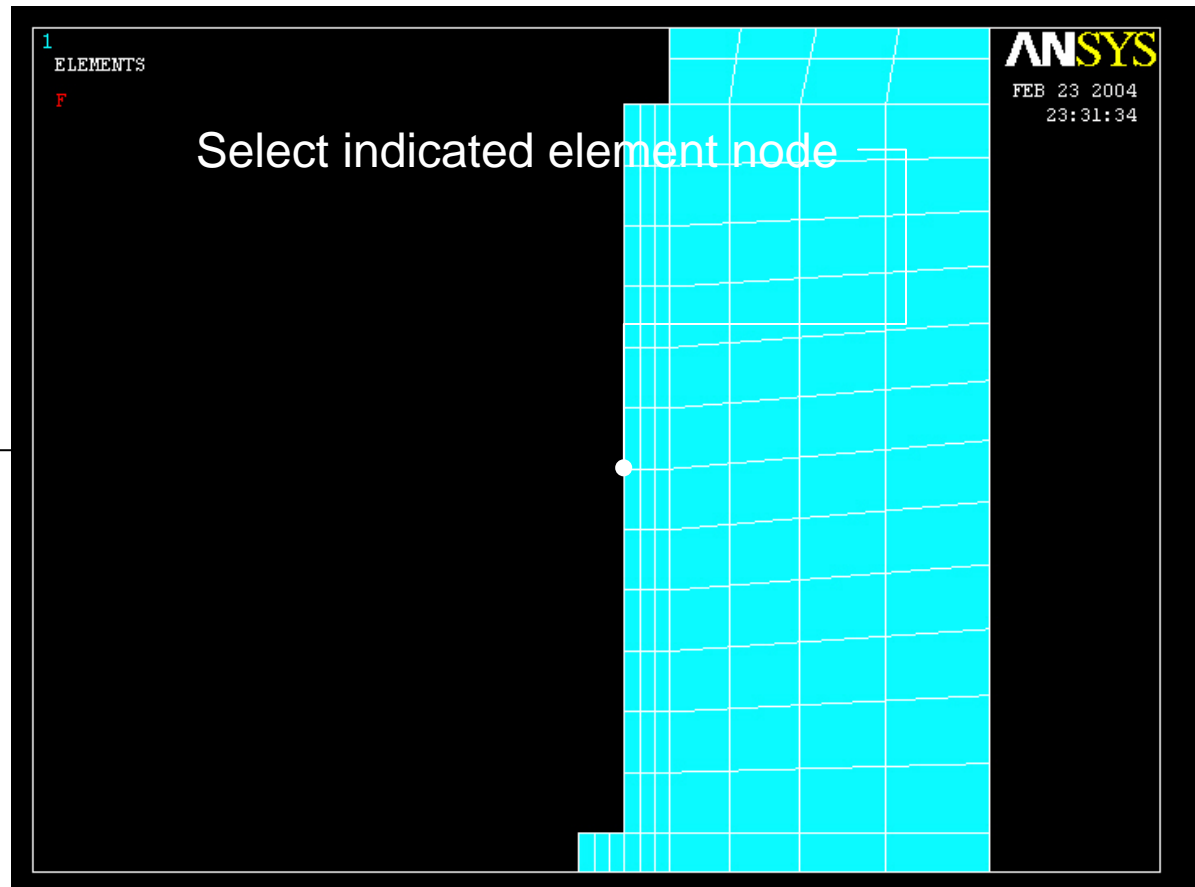
Example – Define Loads

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

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

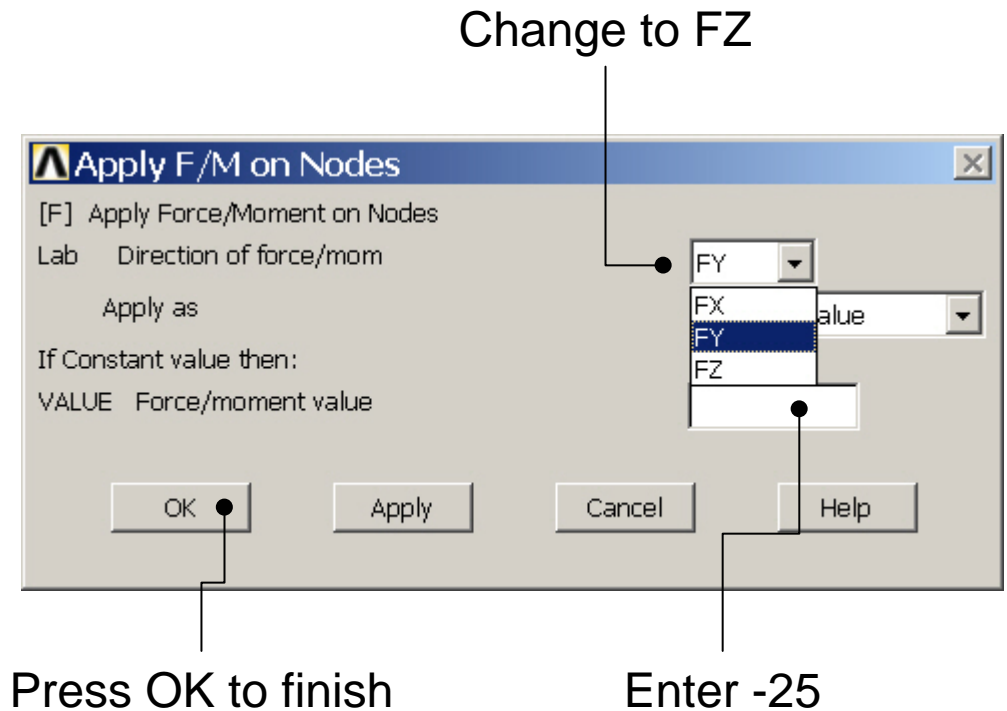


Press OK

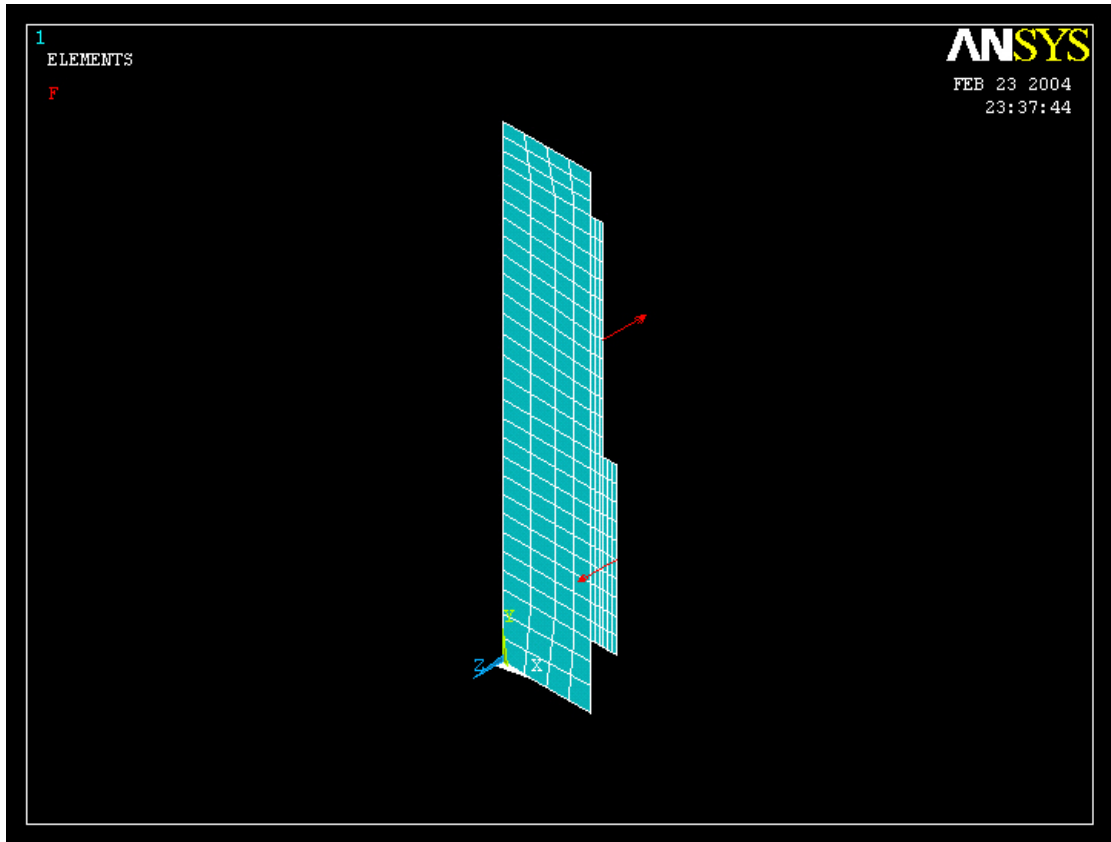


Example – Define Loads

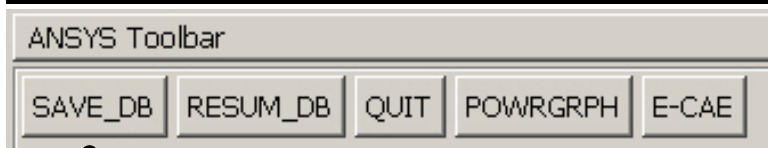
Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes



Example - Save

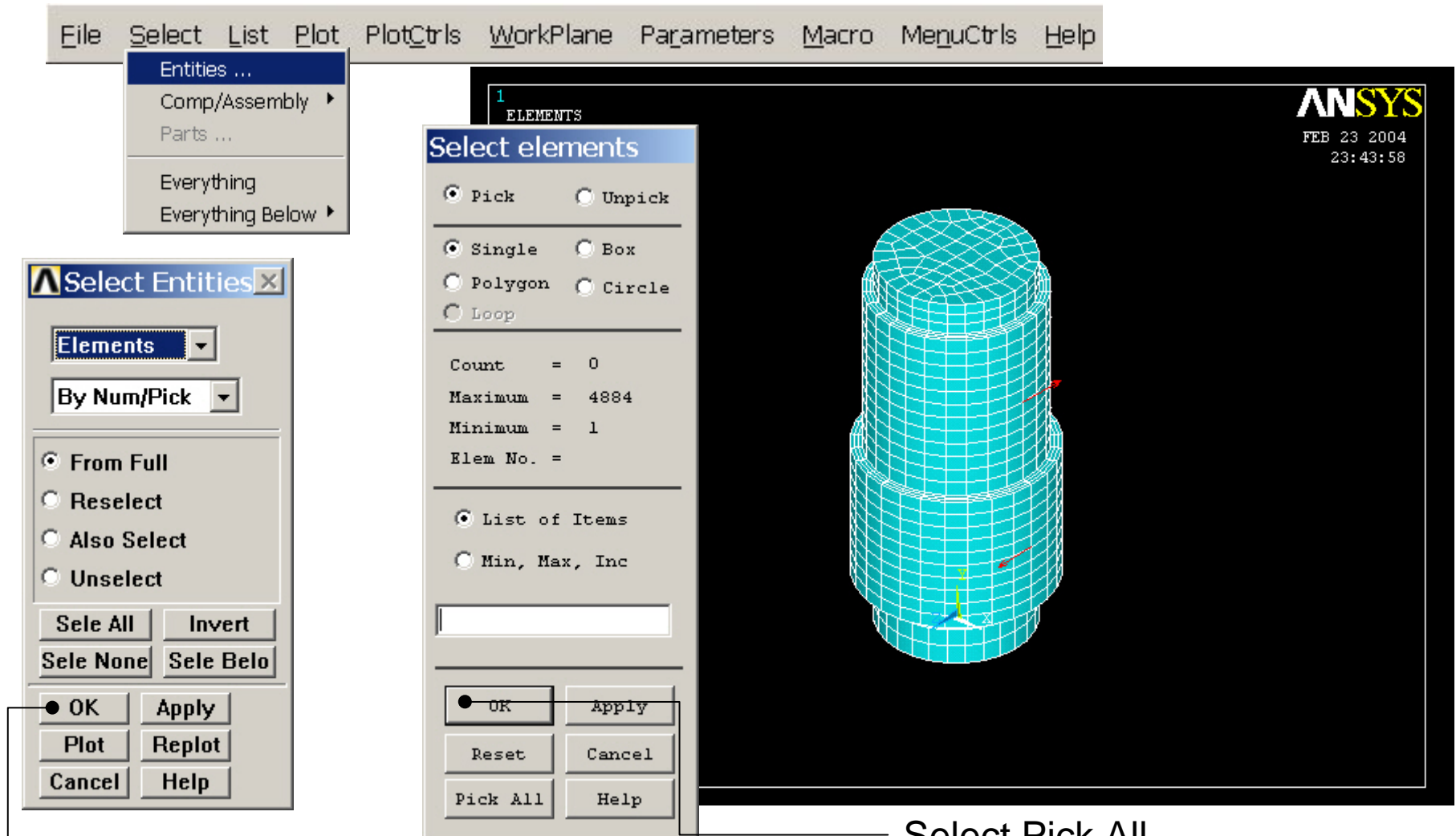


Display of Analysis model

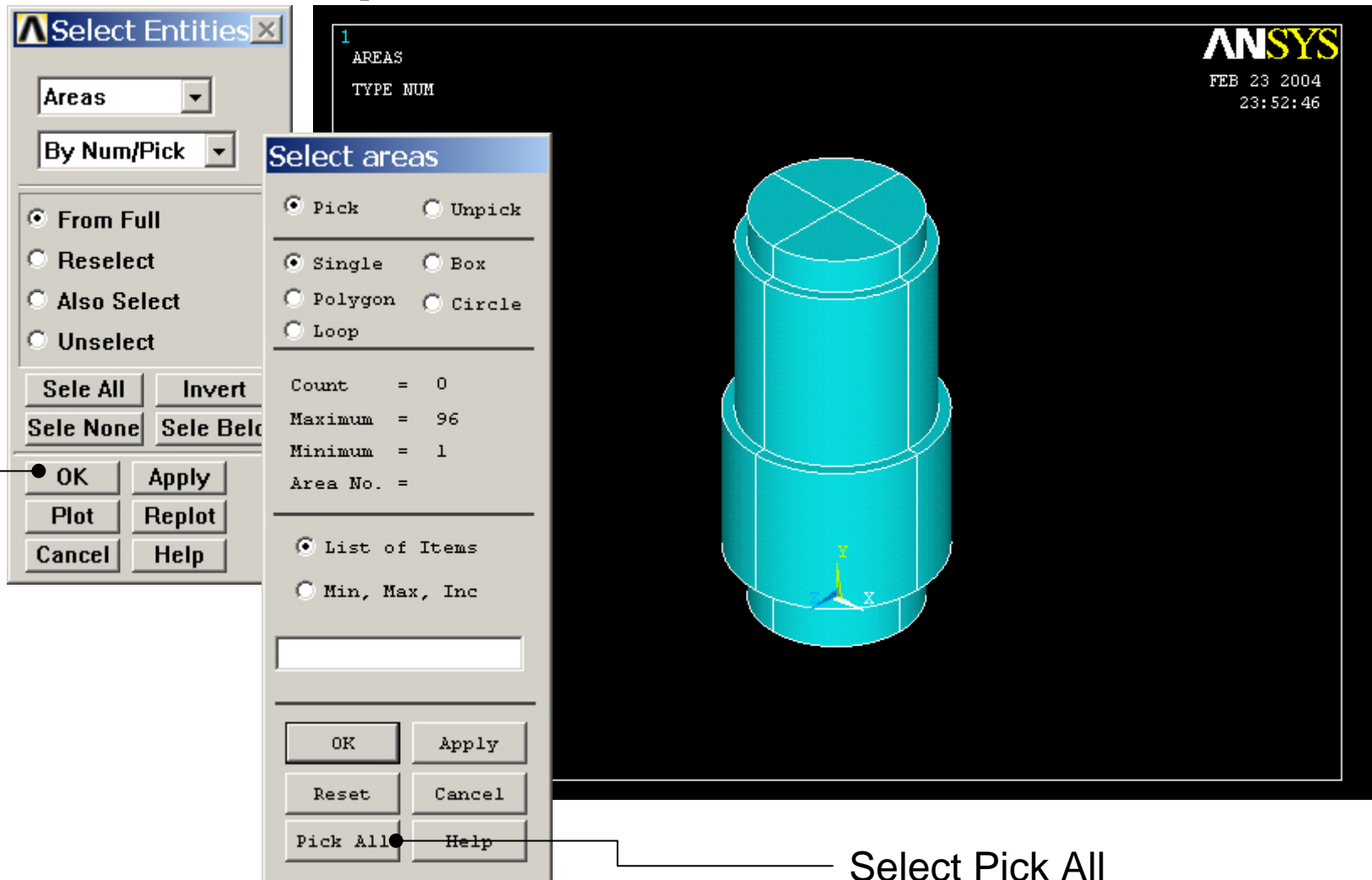


Save the model

Example – Select entities

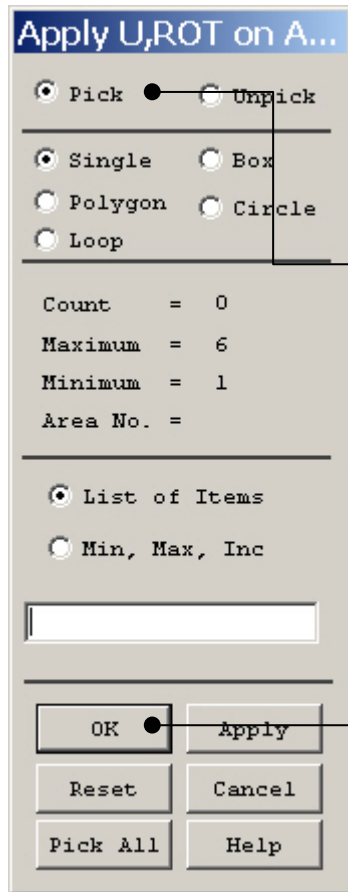


Example – Select entities



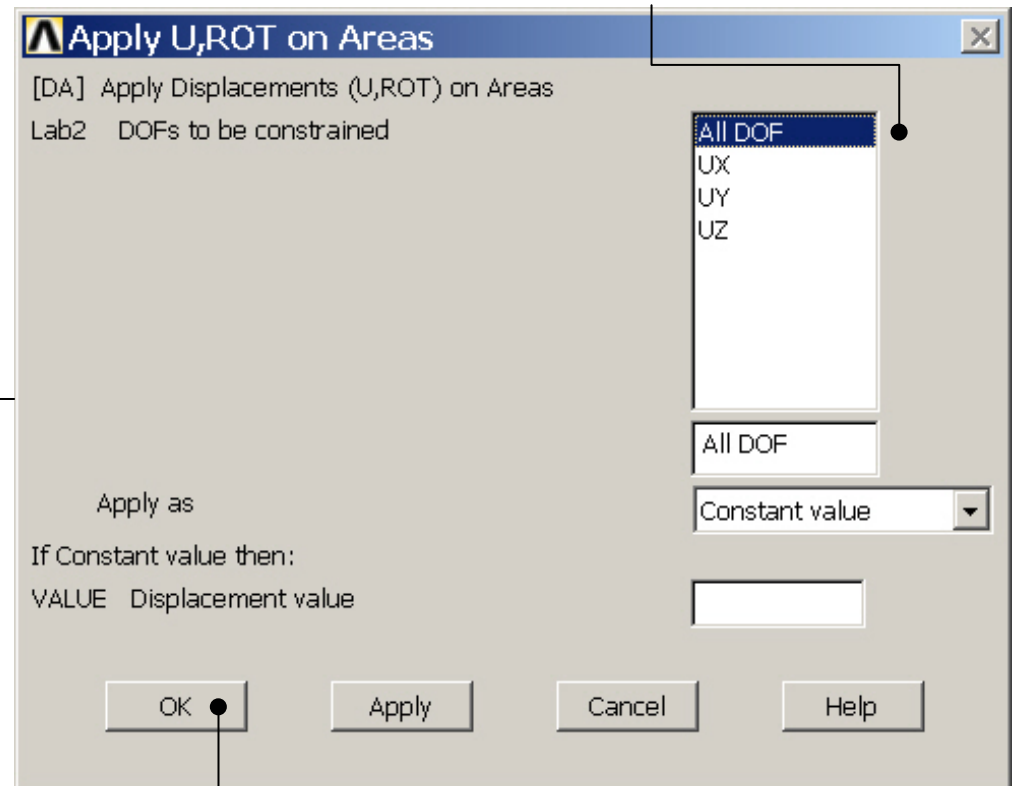
Example – Define Loads

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



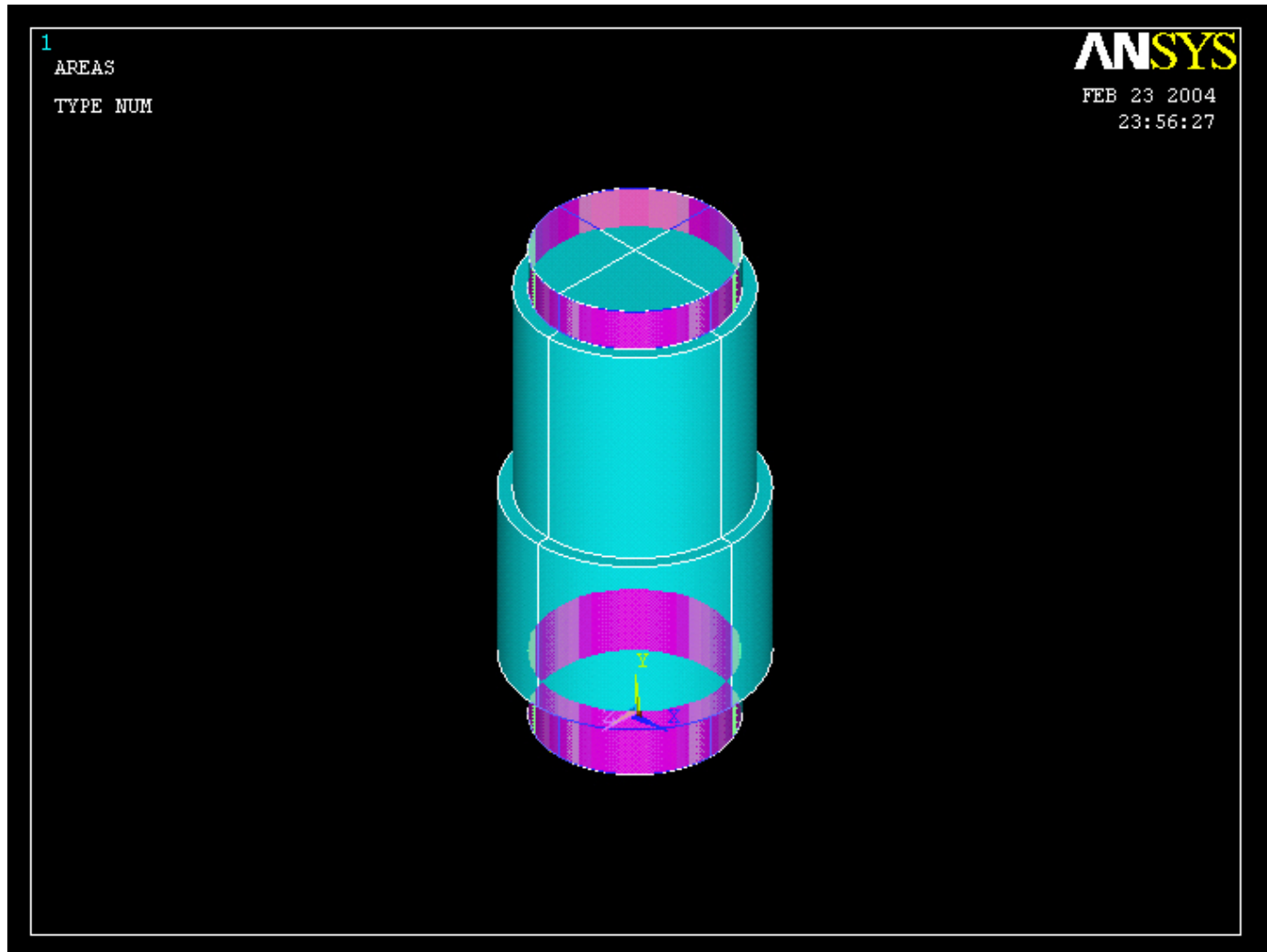
Select Areas
as indicated
on the next
page

Select All DOF to fix/clamp the axle



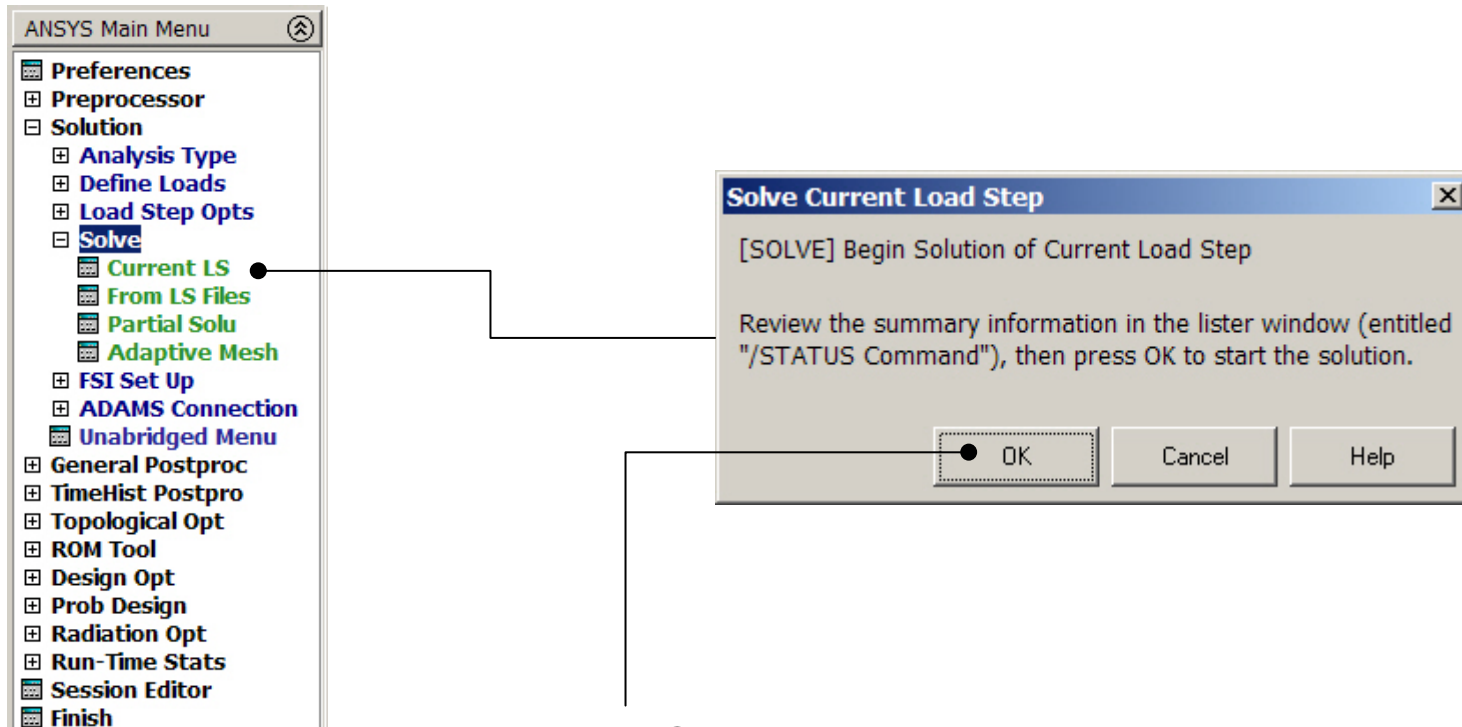
Press OK

Example – Define Loads



Example - Solve

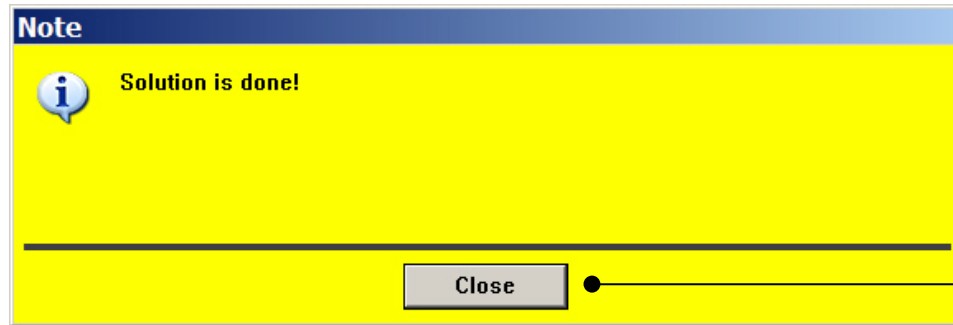
Solution > Solve > Current LS



Press OK

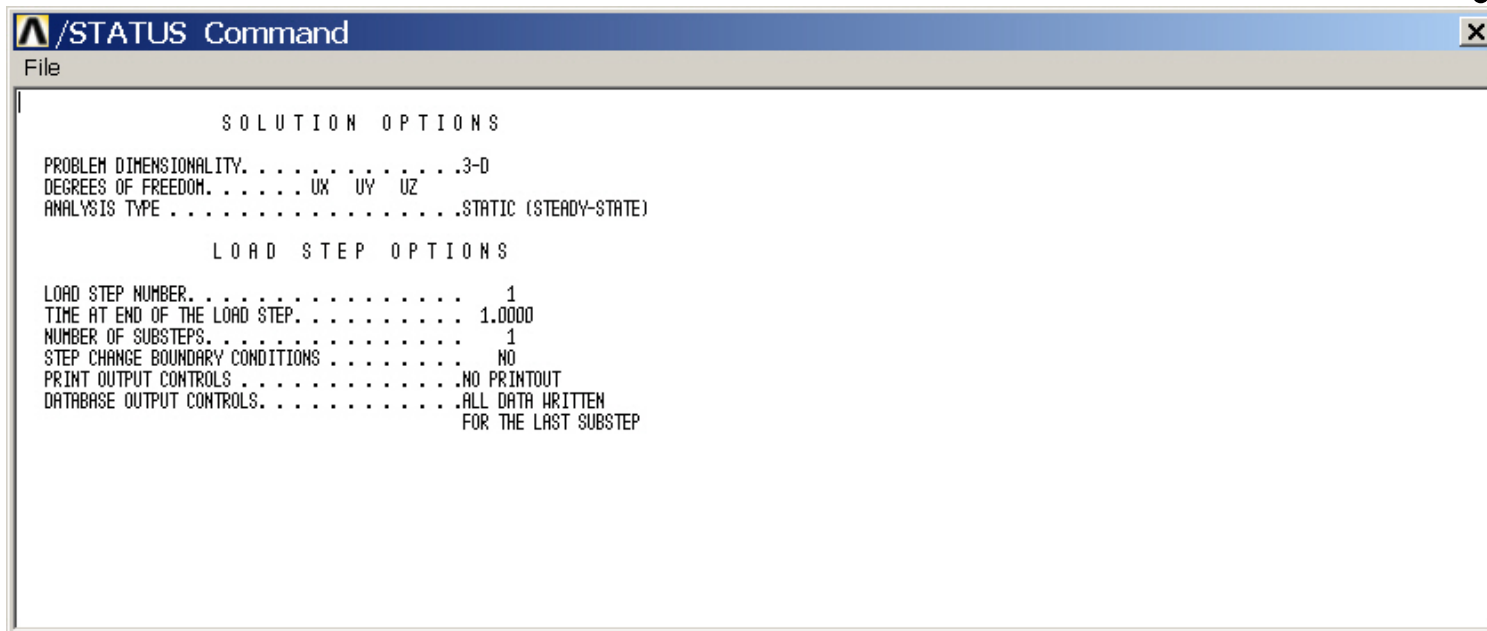
Example0303

Example - Solve



Press Close

Press here
to Close



Example - PostProcessing

The image shows the ANSYS Main Menu on the left and the Contour Nodal Solution Data dialog box on the right. The Main Menu has a tree view with categories: Preferences, Preprocessor, Solution, and General Postproc. Under General Postproc, the 'Plot Results' section is expanded, showing options like Deformed Shape, Contour Plot, Nodal Solu, Element Solu, Elem Table, Line Elem Res, Vector Plot, Plot Path Item, Concrete Plot, List Results, Query Results, Options for Outp, Results Viewer, Write PGR File, Nodal Calcs, Element Table, Path Operations, Load Case, Check Elem Shape, Write Results, ROM Operations, Submodeling, and Fatigue. The 'Contour Plot' option is selected. The Contour Nodal Solution Data dialog box has a title bar and a close button. It contains the following fields and options: [PLNSOL] Contour Nodal Solution Data, Item,Comp Item to be contoured (with a list of options: DOF solution, Stress, Strain-total, Strain-mech+thrm, Energy, Strain ener dens, Strain-elastic), 3rd principal S3, Intensity SINT, von Mises SEQV (selected), PlasEqvStrs SEPL, StressRatio SRAT, von Mises SEQV (in a separate box), KUND Items to be plotted (with radio buttons: Def shape only, Def + undeformed (selected), Def + undef edge), Fact Optional scale factor (with a text box containing 1), [/EFACET] Interpolation Nodes (with radio buttons: Corner only (selected), Corner + midside, All applicable), [AVPRIN] Eff NU for EQV strain (with a text box), and buttons for OK, Apply, Cancel, and Help. A text box on the right says 'Select "Def+undeformed" and Press OK' with arrows pointing to the 'Def + undeformed' radio button and the OK button.

ANSYS Main Menu

- Preferences
- Preprocessor
- Solution
- General Postproc
 - Data & File Opts
 - Results Summary
 - Read Results
 - Failure Criteria
 - Plot Results
 - Deformed Shape
 - Contour Plot
 - Nodal Solu
 - Element Solu
 - Elem Table
 - Line Elem Res
 - Vector Plot
 - Plot Path Item
 - Concrete Plot
- List Results
- Query Results
- Options for Outp
- Results Viewer
- Write PGR File
- Nodal Calcs
- Element Table
- Path Operations
- Load Case
- Check Elem Shape
- Write Results
- ROM Operations
- Submodeling
- Fatigue

Contour Nodal Solution Data

[PLNSOL] Contour Nodal Solution Data

Item,Comp Item to be contoured

DOF solution
Stress
Strain-total
Strain-mech+thrm
Energy
Strain ener dens
Strain-elastic

3rd principal S3
Intensity SINT
von Mises SEQV
PlasEqvStrs SEPL
StressRatio SRAT

von Mises SEQV

KUND Items to be plotted

☐ Def shape only
☒ Def + undeformed
☐ Def + undef edge

Fact Optional scale factor

1

[/EFACET] Interpolation Nodes

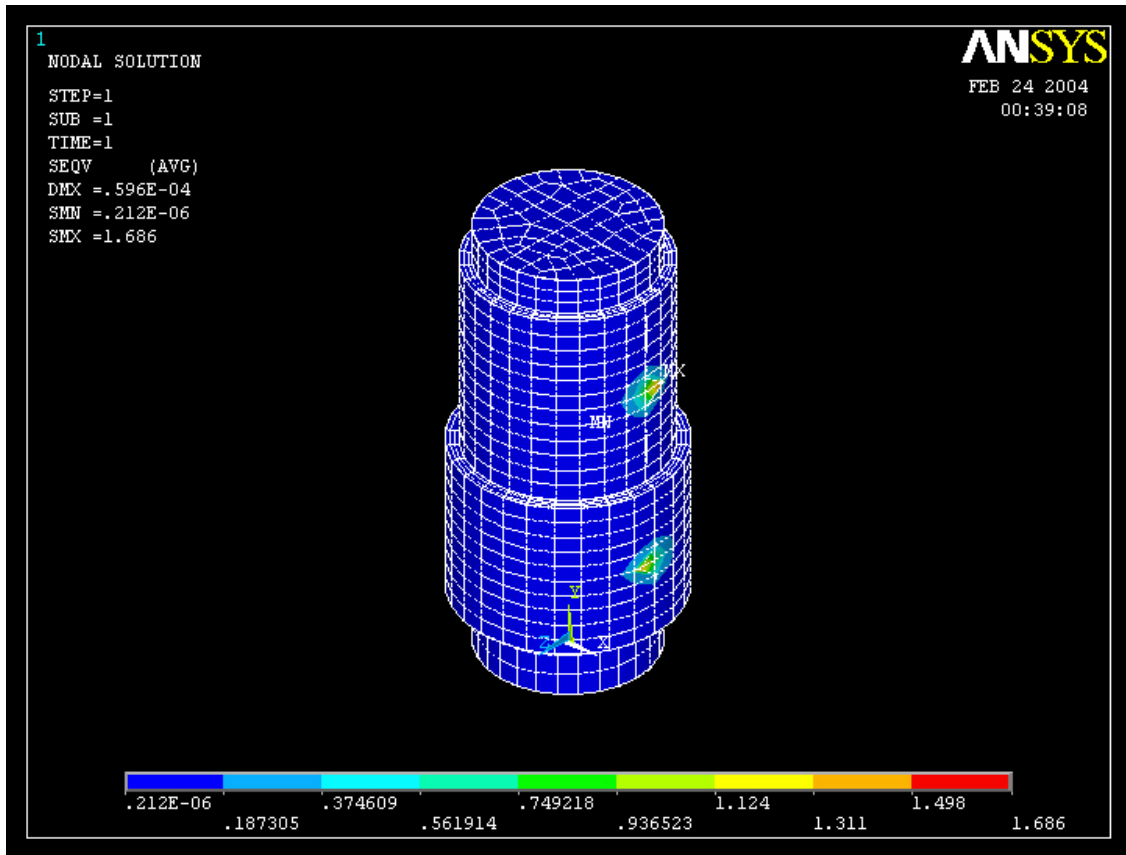
☒ Corner only
☐ Corner + midside
☐ All applicable

[AVPRIN] Eff NU for EQV strain

OK Apply Cancel Help

Select "Def+undeformed" and Press OK

Example - PostProcessing



Read Maximum displacement: DMX