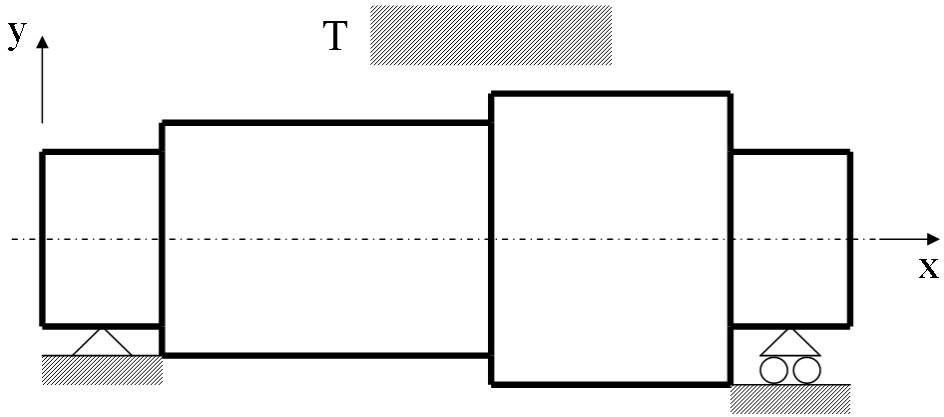
Course in ANSYS

Example0303

Example – Gear axle 3D



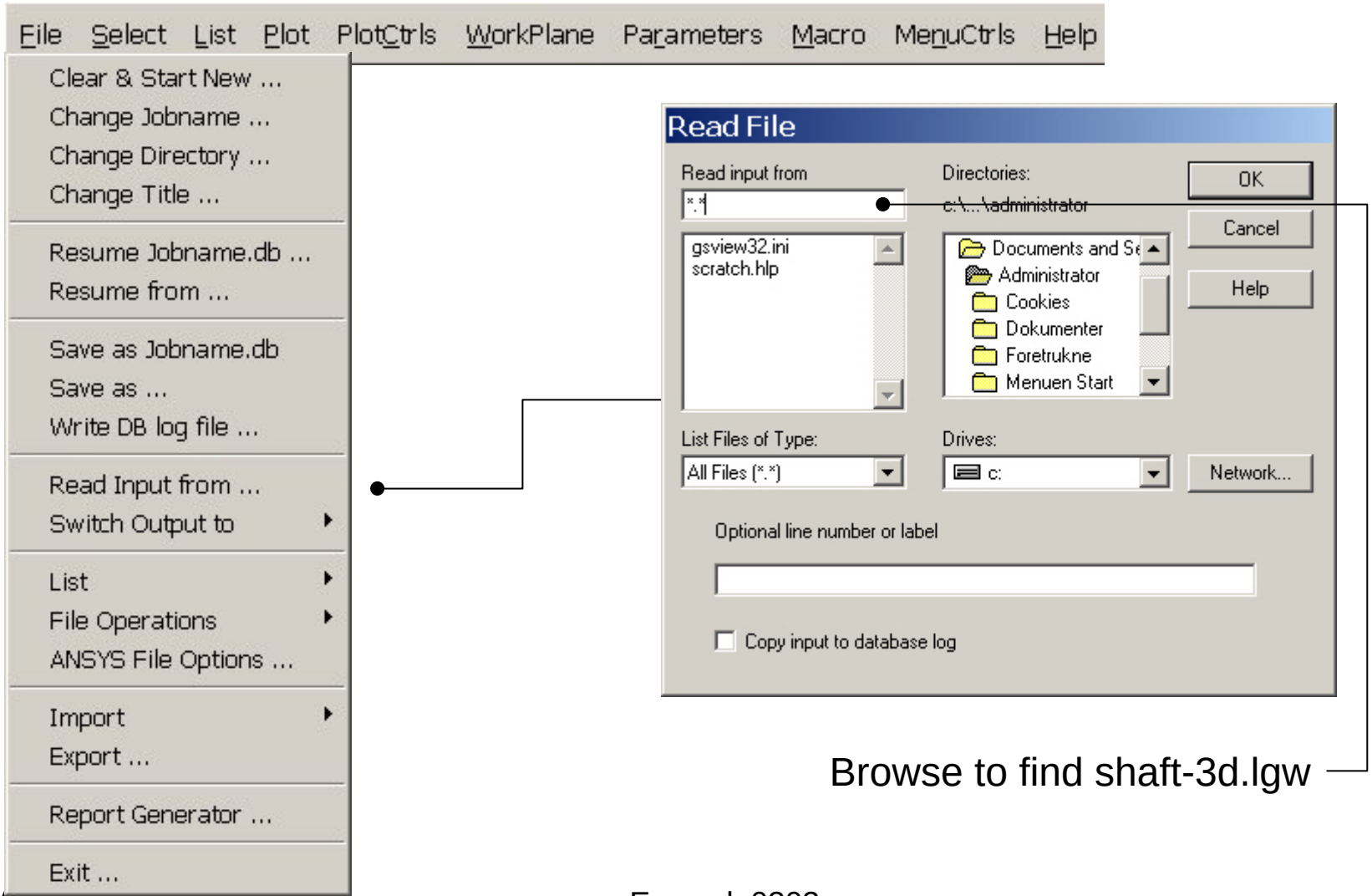
$E = 210000 N/mm^2$

$\nu = 0.3$

$T = 1000 Nmm$

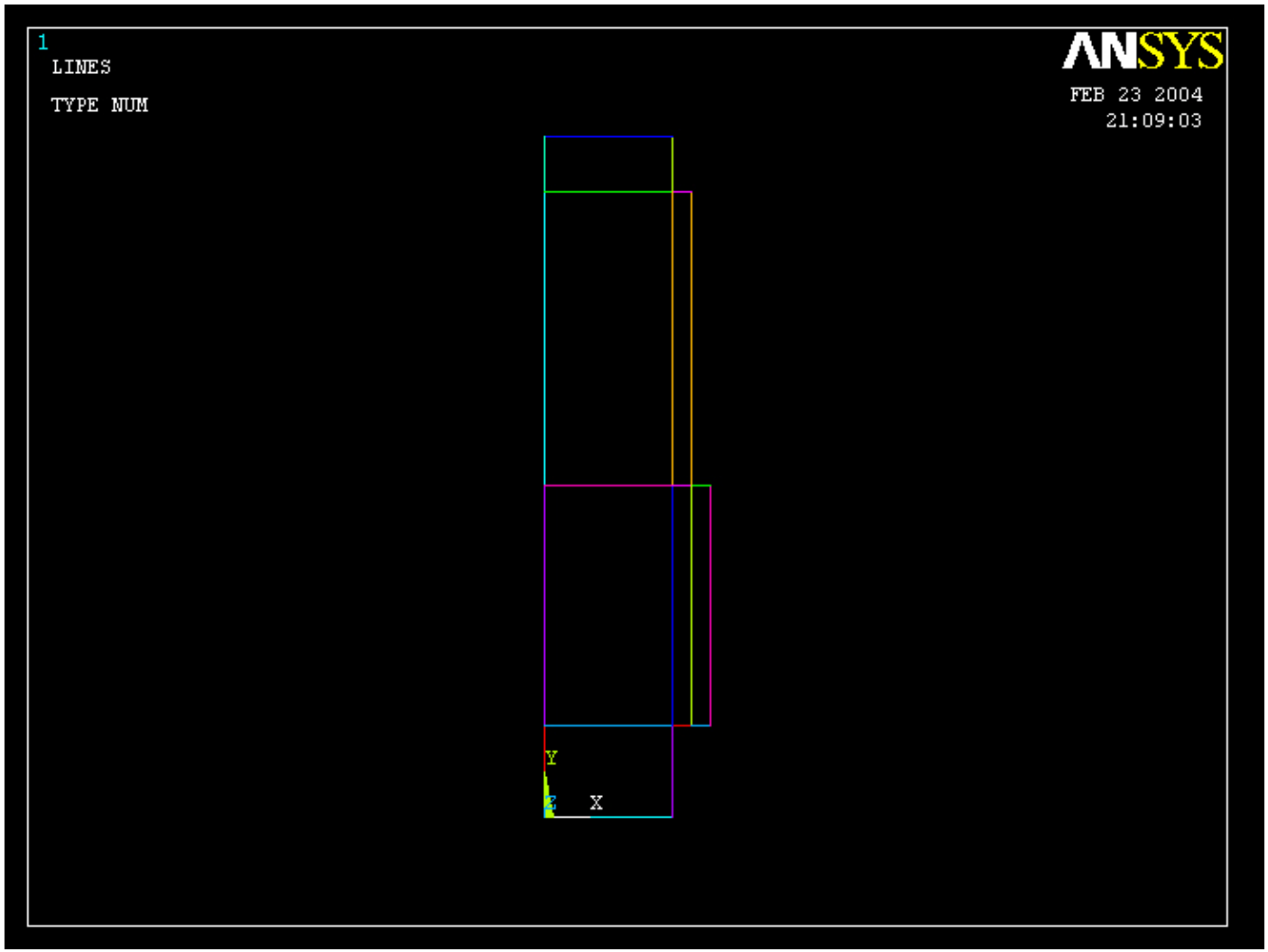
Objective:
Compute the maximum stress von Mise
Tasks:
How should this be modeled?
Topics:
Element type, Real constants, modeling,
Plot results, output graphics, select entities

Example – Read Input from

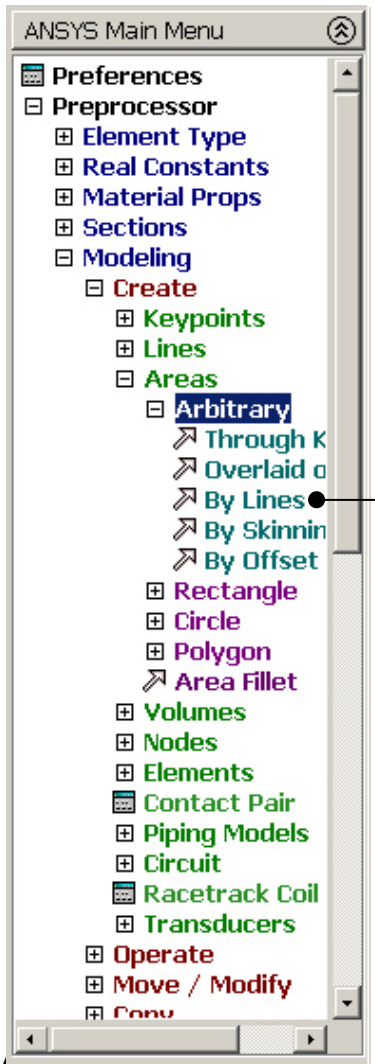


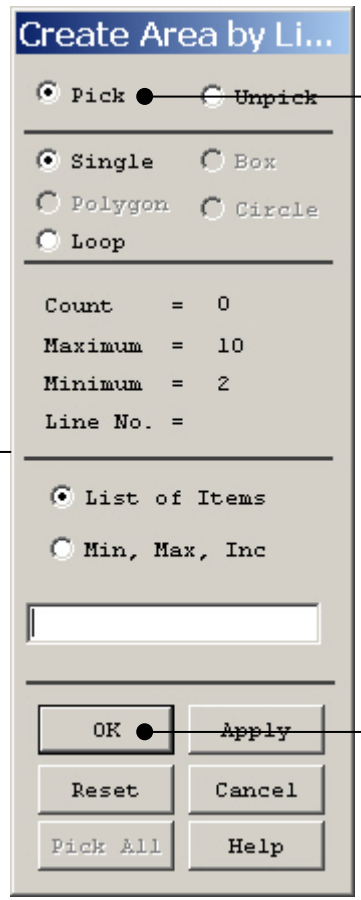
Browse to find shaft-3d.lgw

Example – Read Input from



Example – Create Areas by line

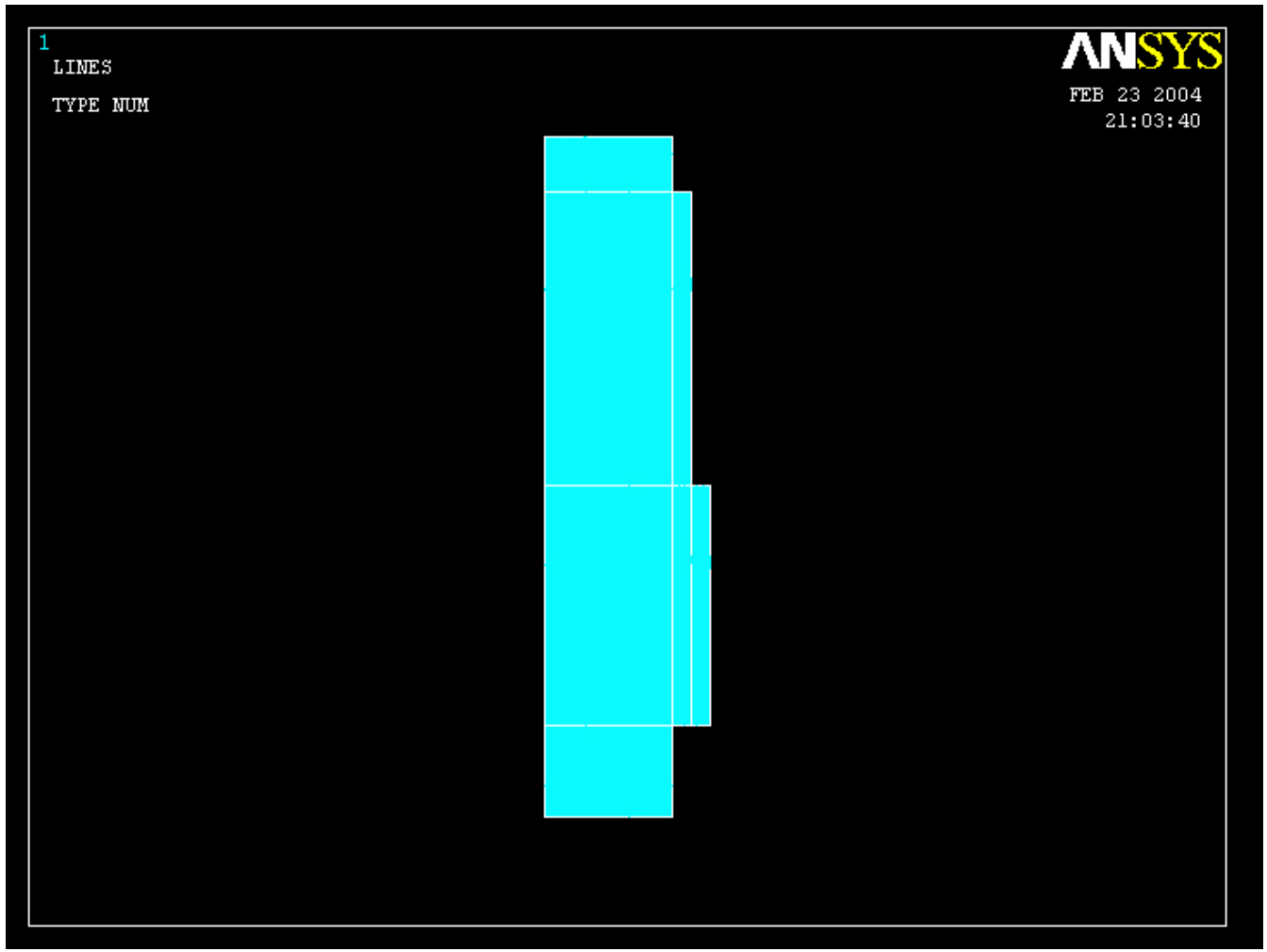




Select All lines

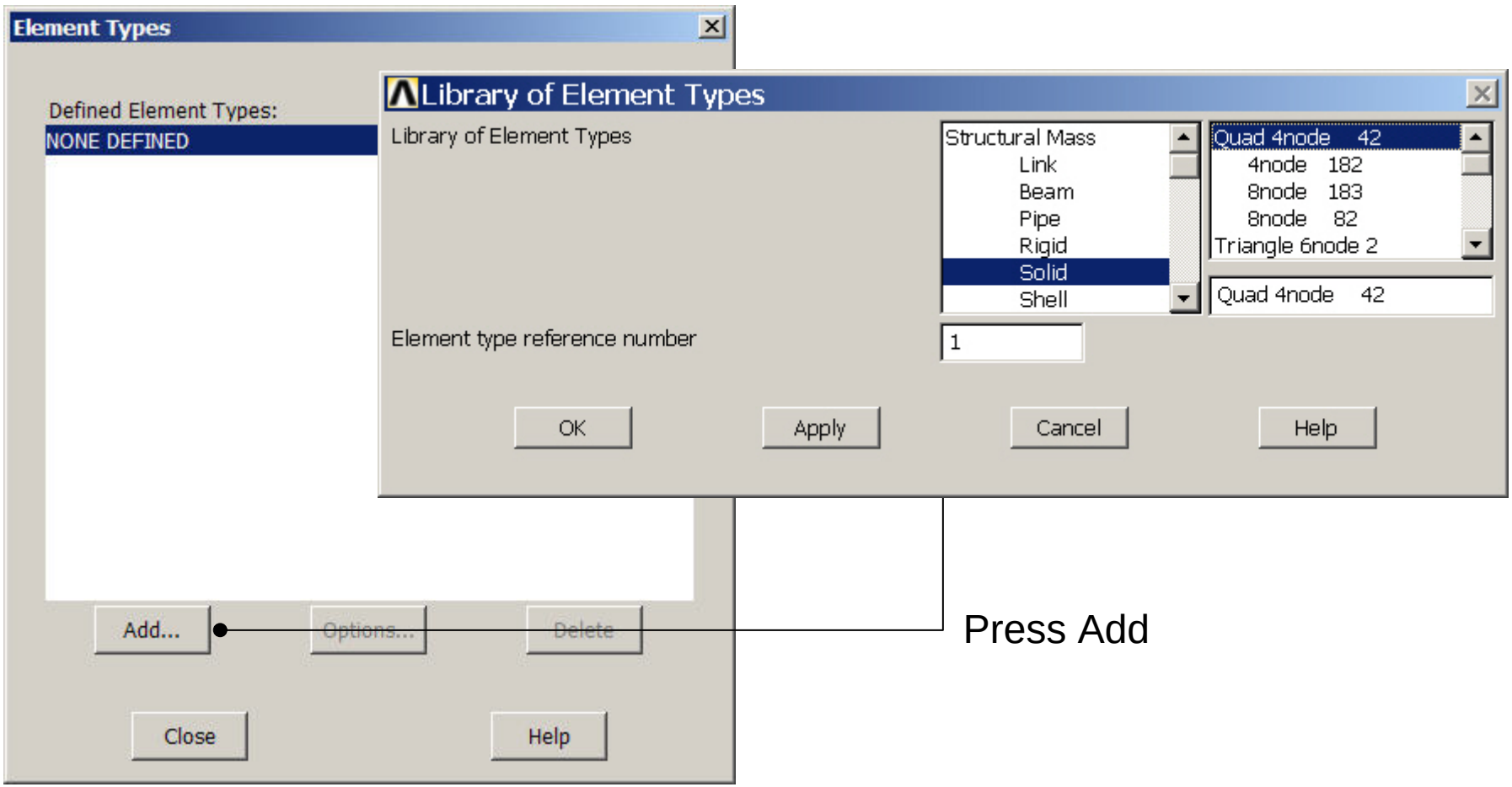
Press OK to finish

Example – Create Areas by line



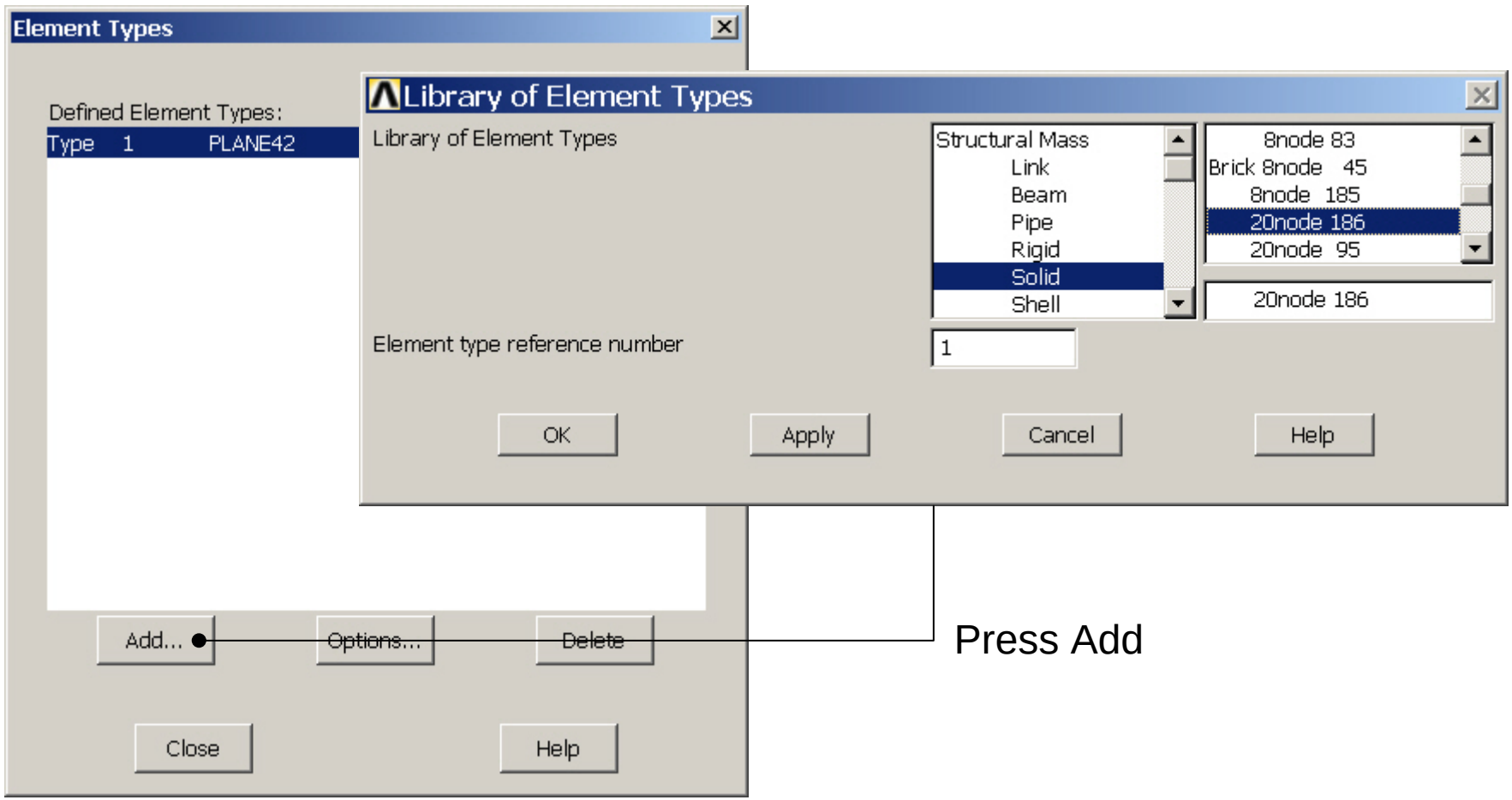
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete

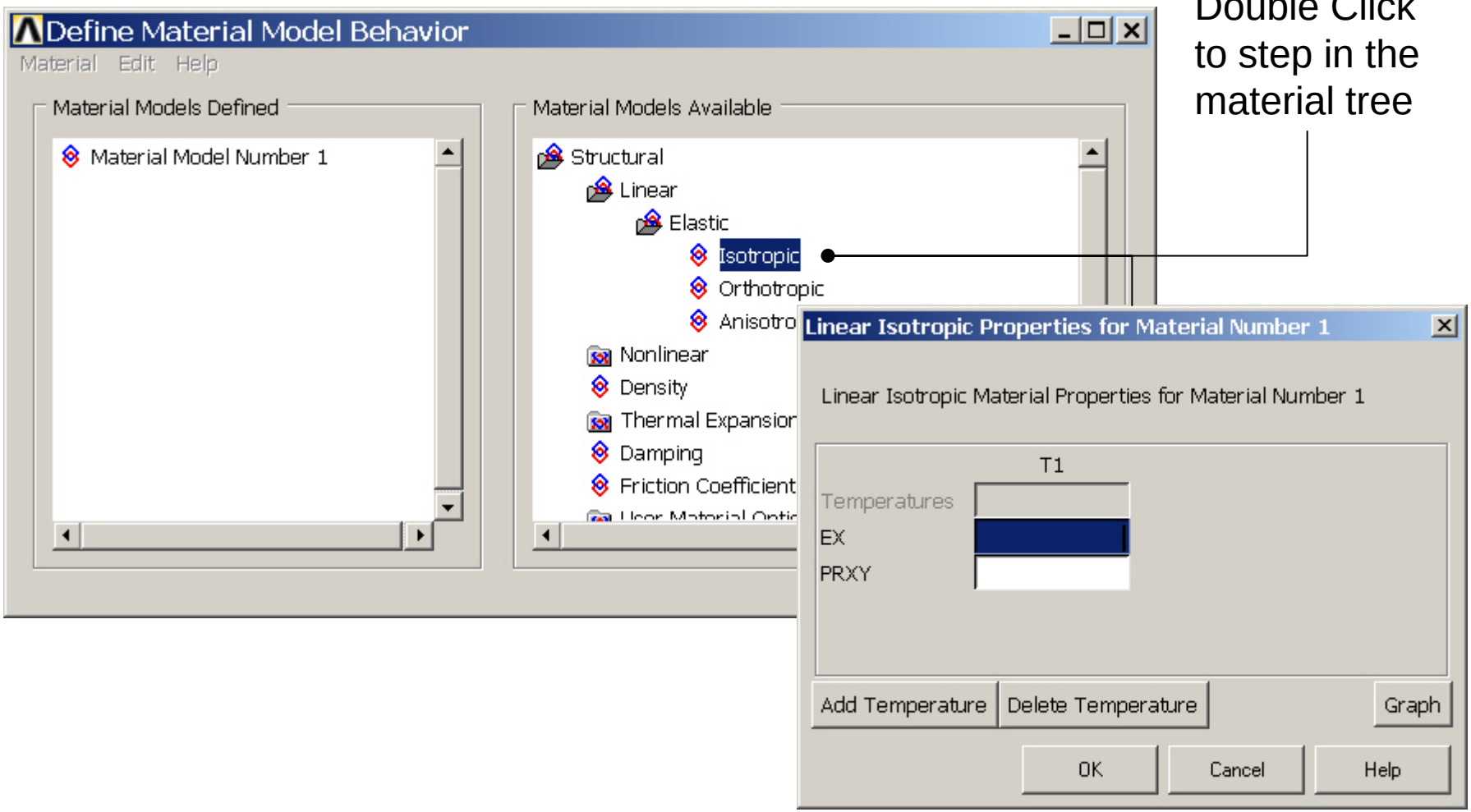


Example – Real Constants

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

Example - Material Properties

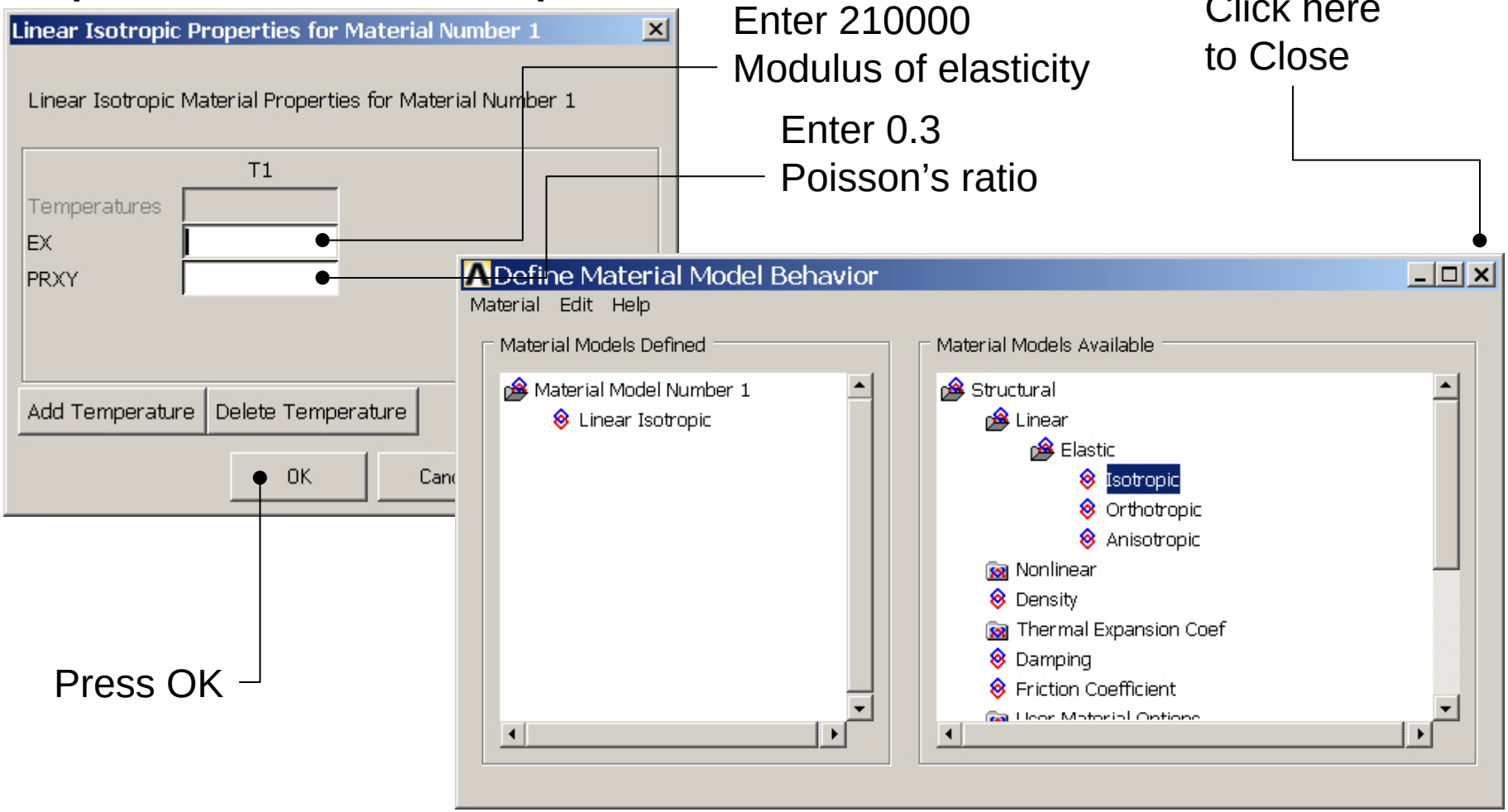
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

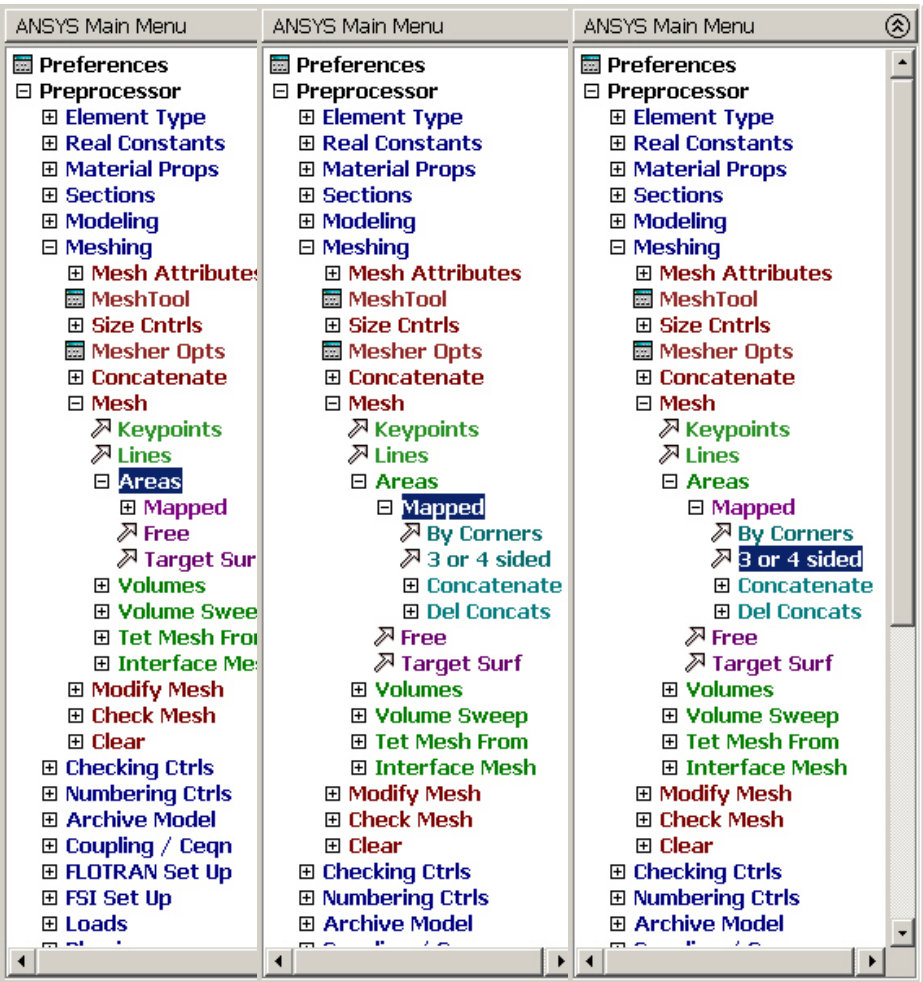
Example - Material Properties

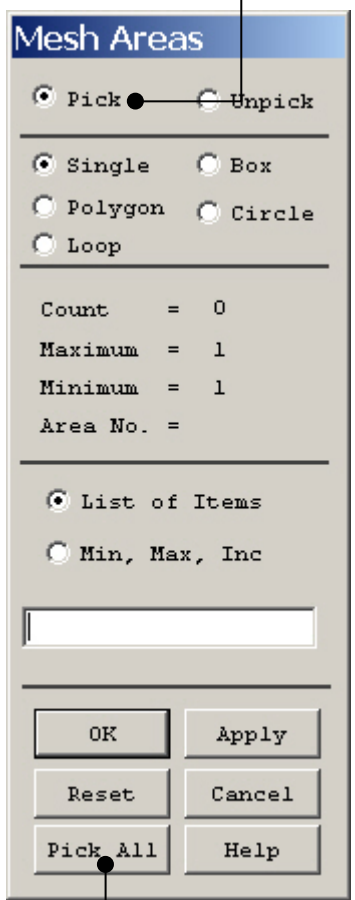
Preprocessor > Material Props > Material Models



Example - Meshing

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



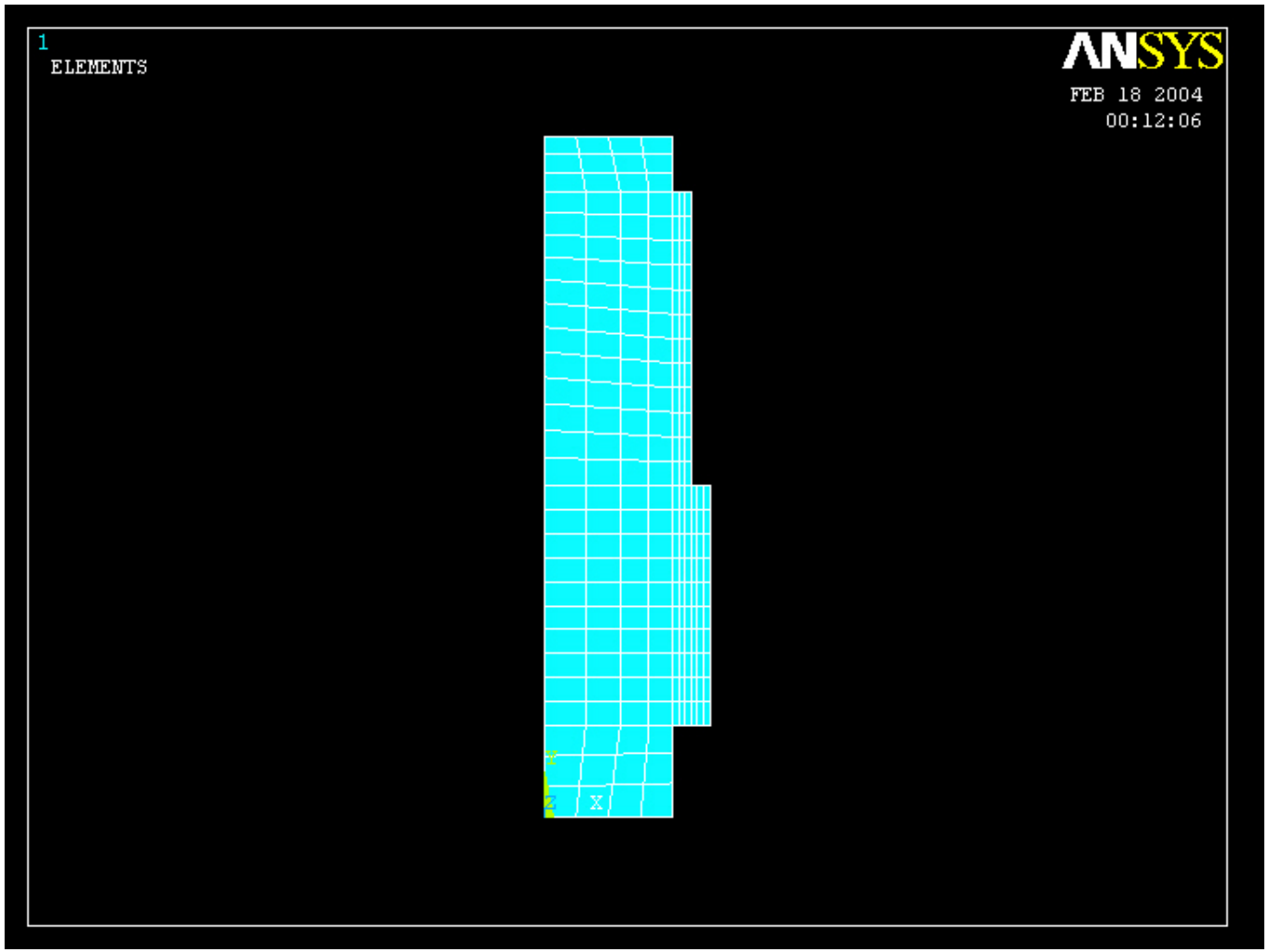


Select individual areas to be meshed

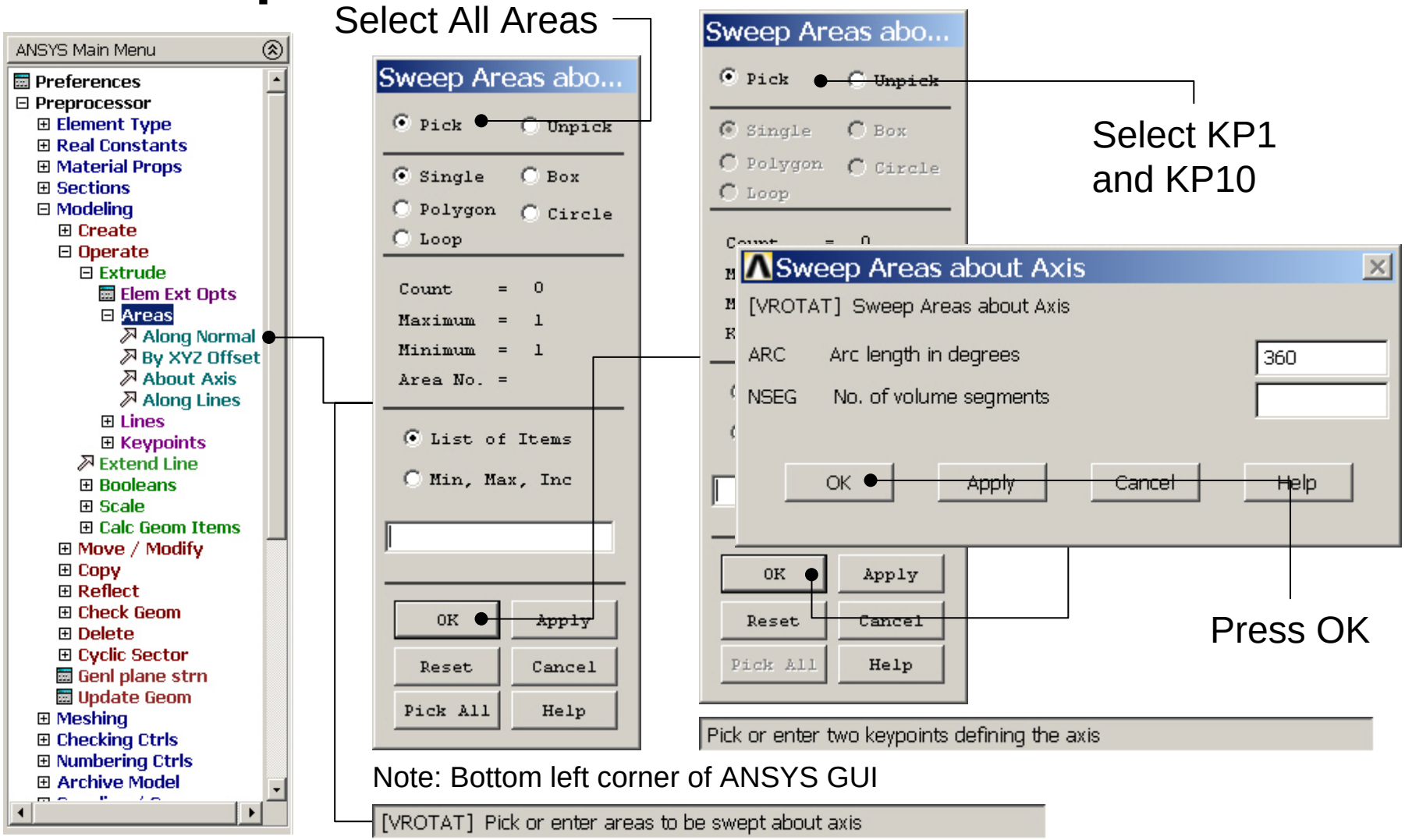
NB: It is often necessary to “Clear” the model for example if Element Type or model geometry is to be changed

Select all areas defined to be meshed

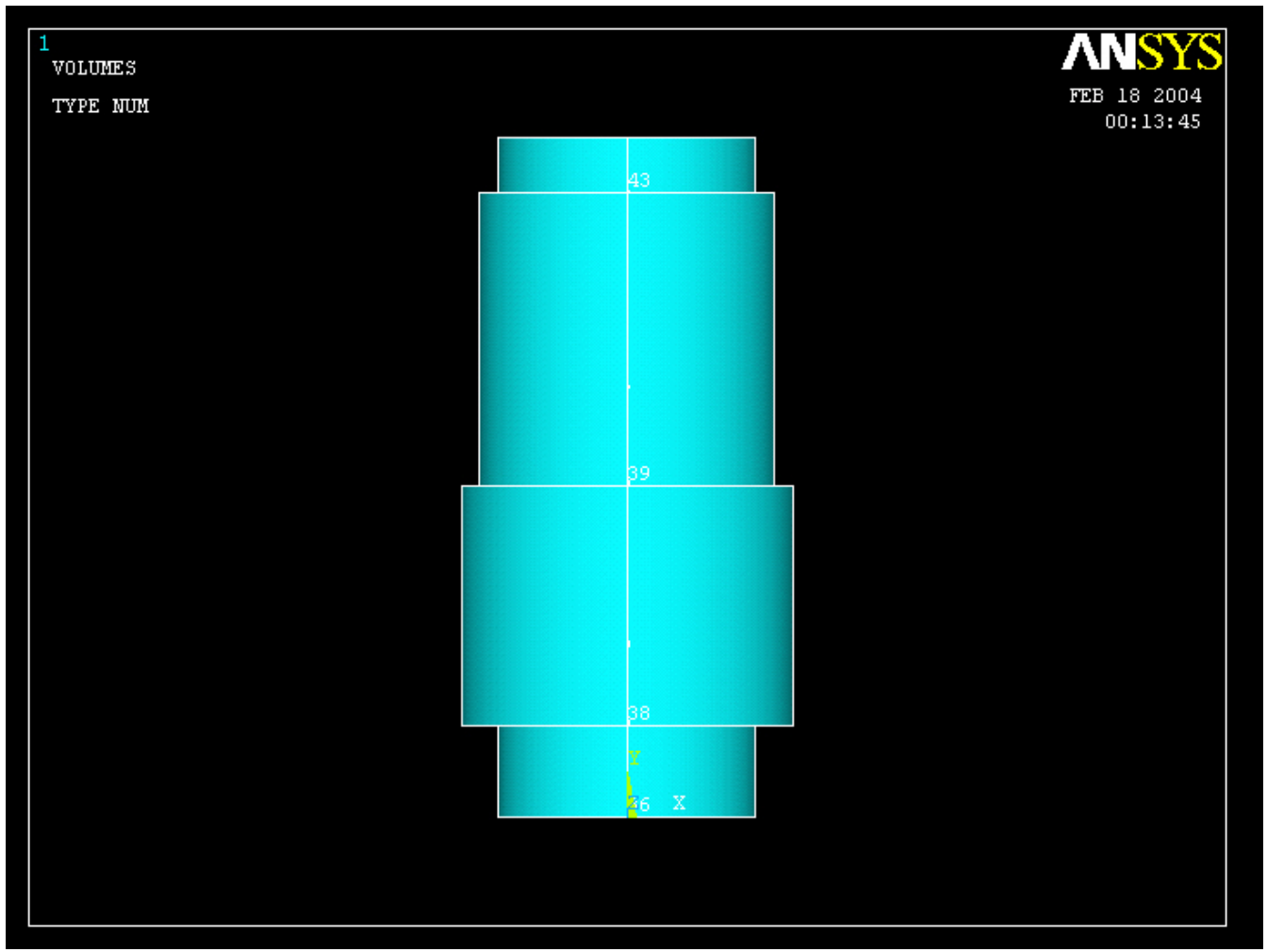
Example - Meshing



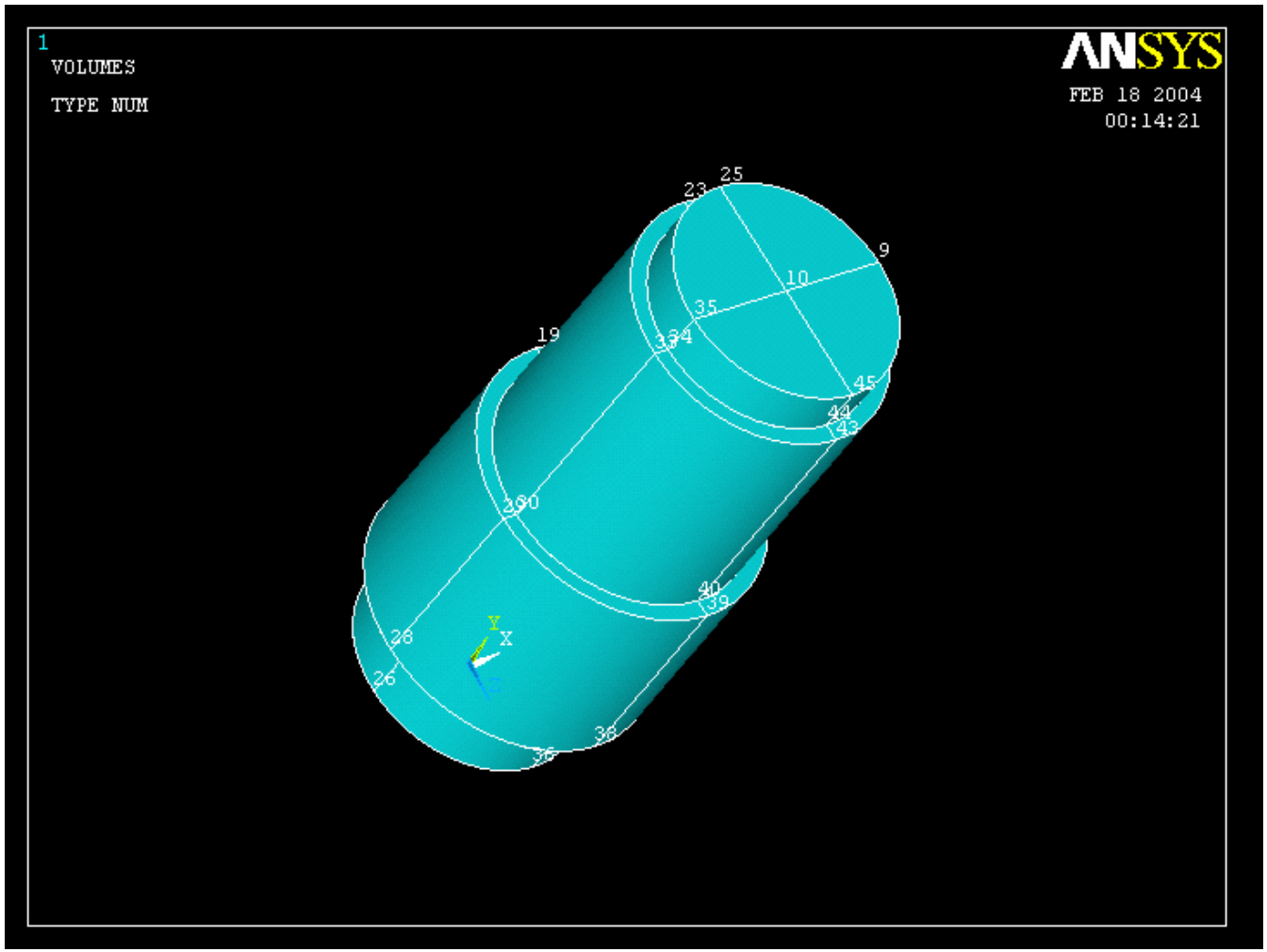
Example – Extrude – About Axis



Example – Extrude – About Axis

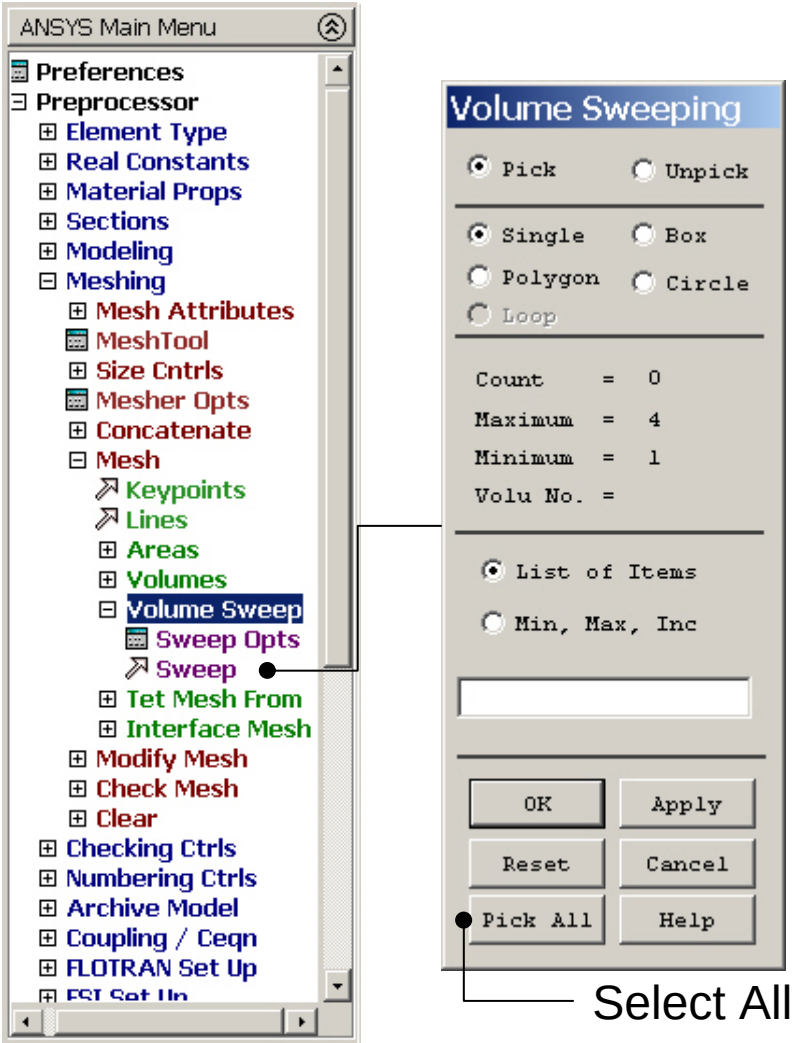


Example – Mouse rotate

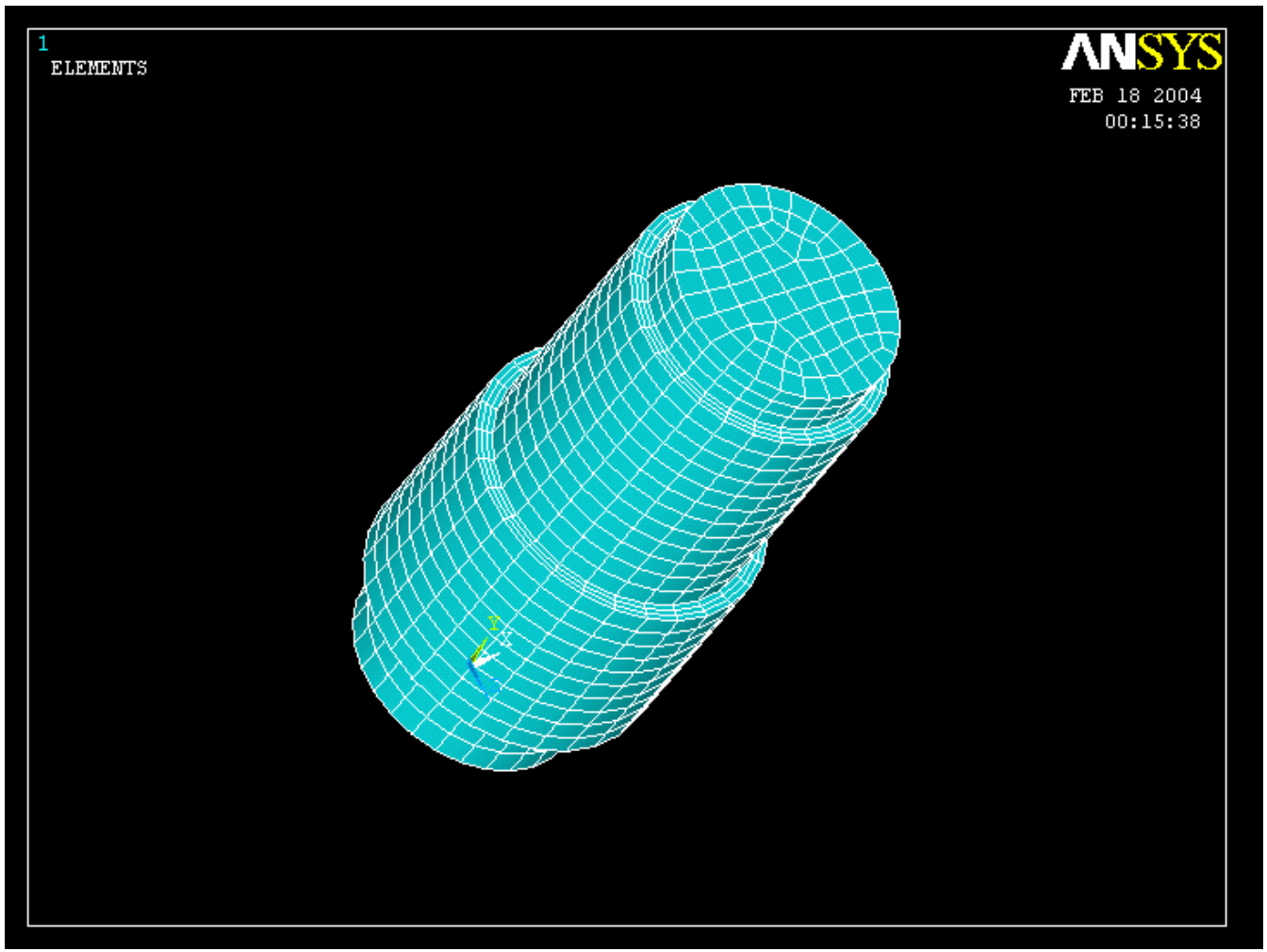


Rotate by holding the Ctrl key down while using the right hand mouse button

Example – Volume Sweep



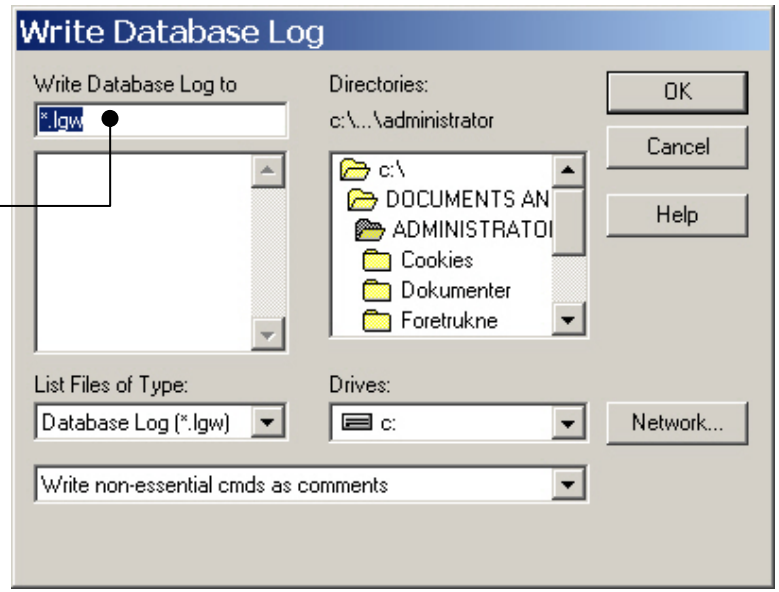
Example – 3D Mesh



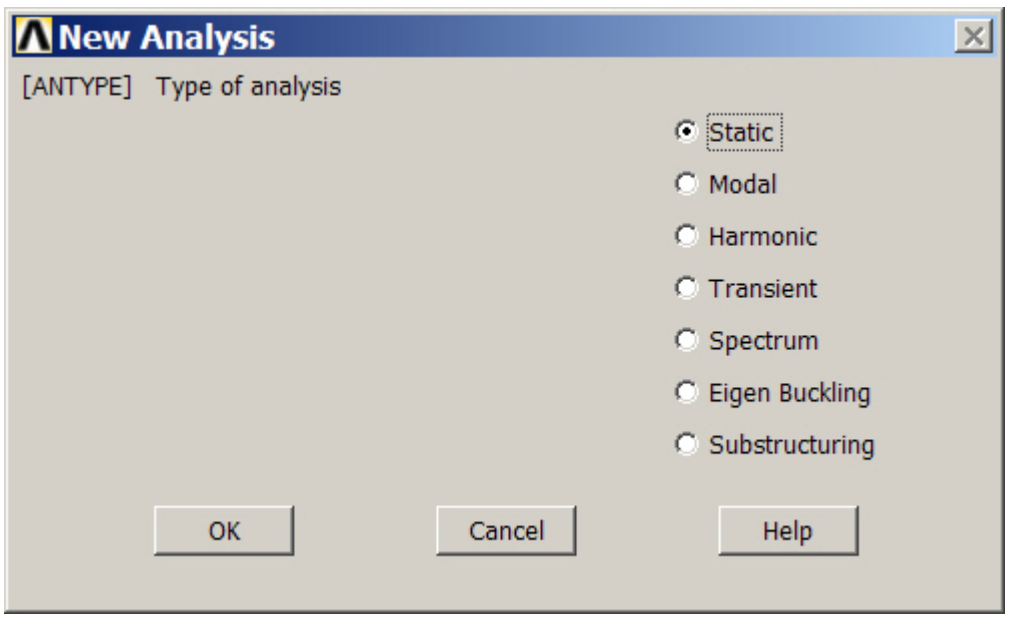
Example – Analysis Type

File > Write DB log file

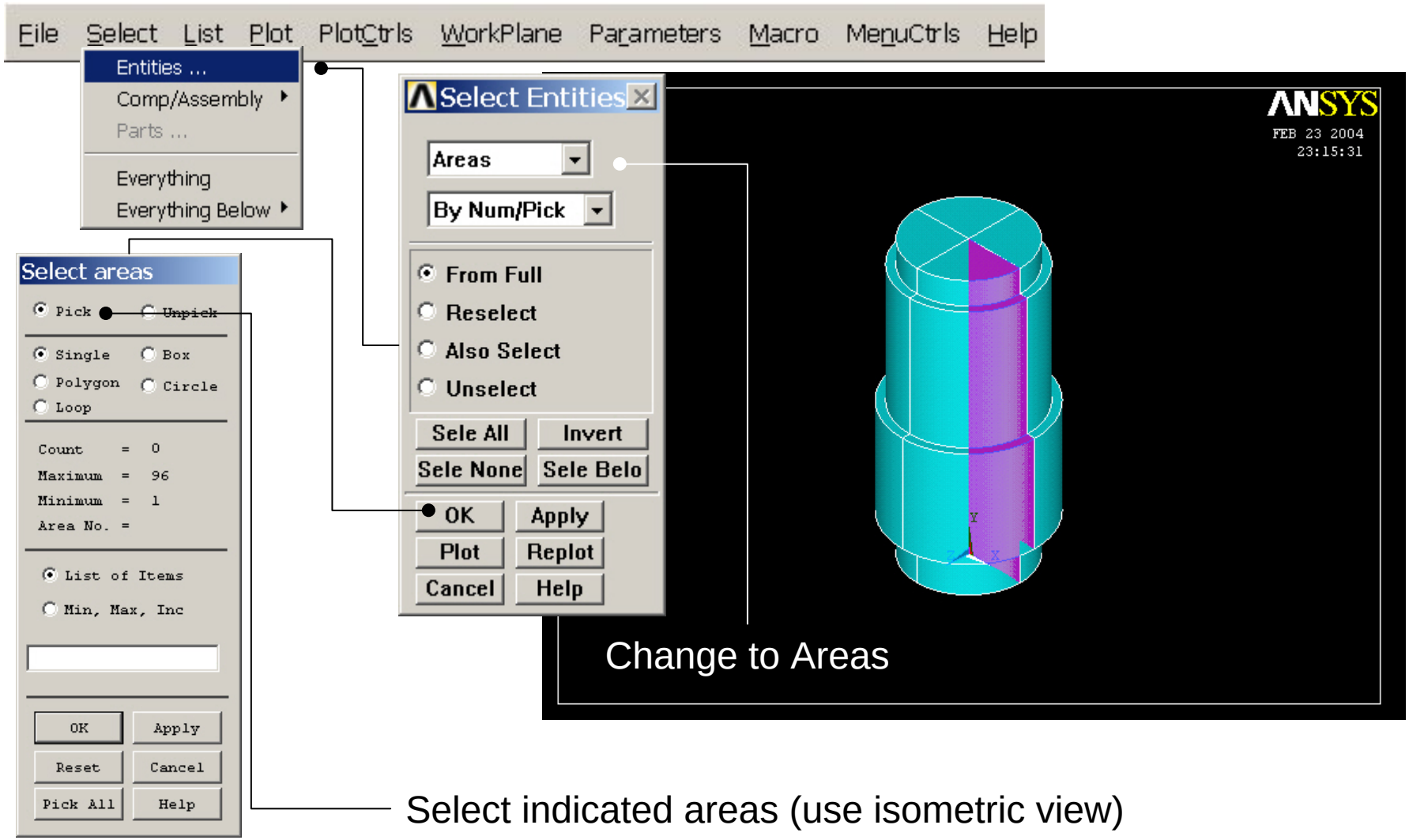
Enter “example0303.lgw”



Solution > Analysis Type > New Analysis

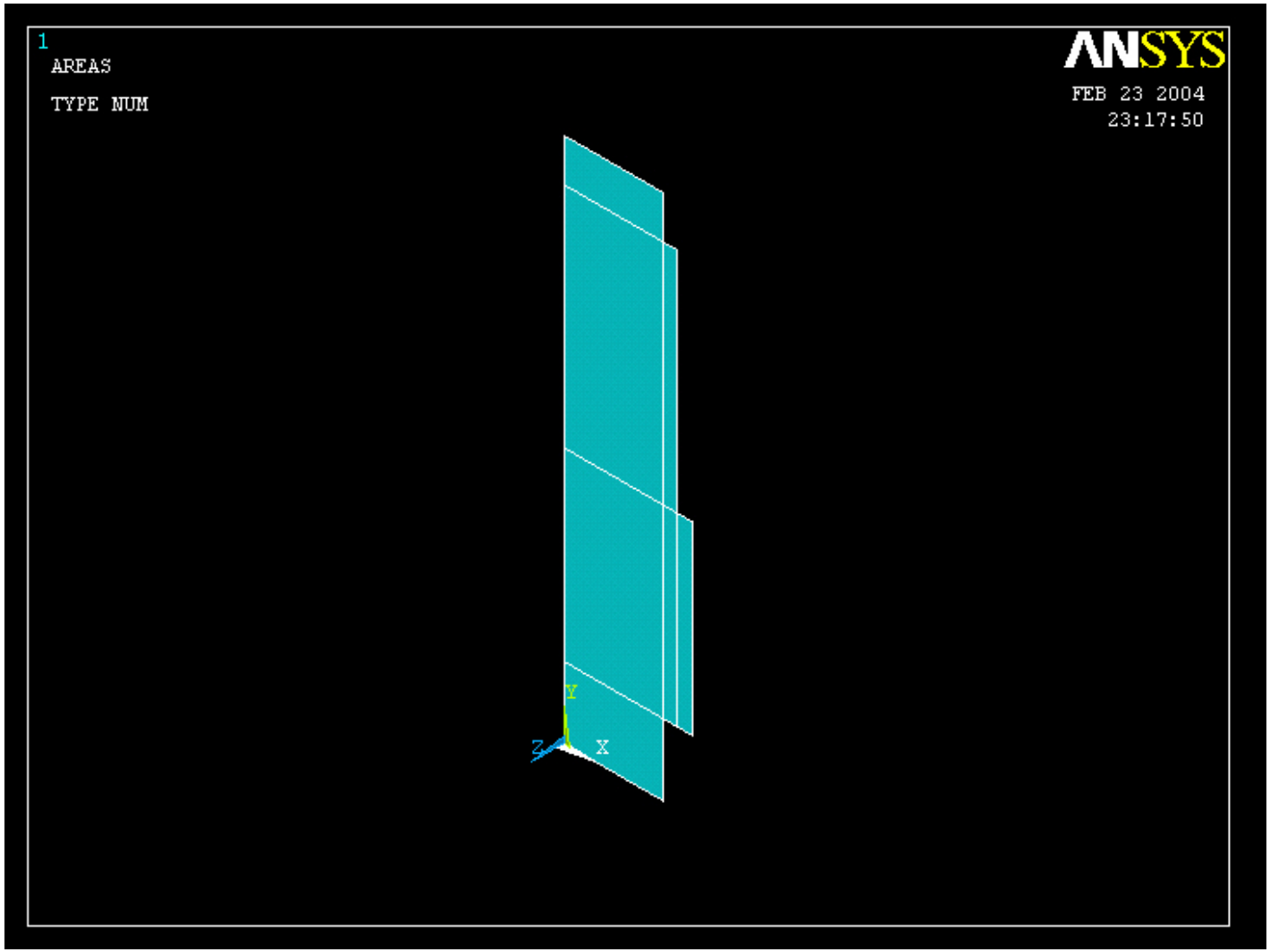


Example – Select entities

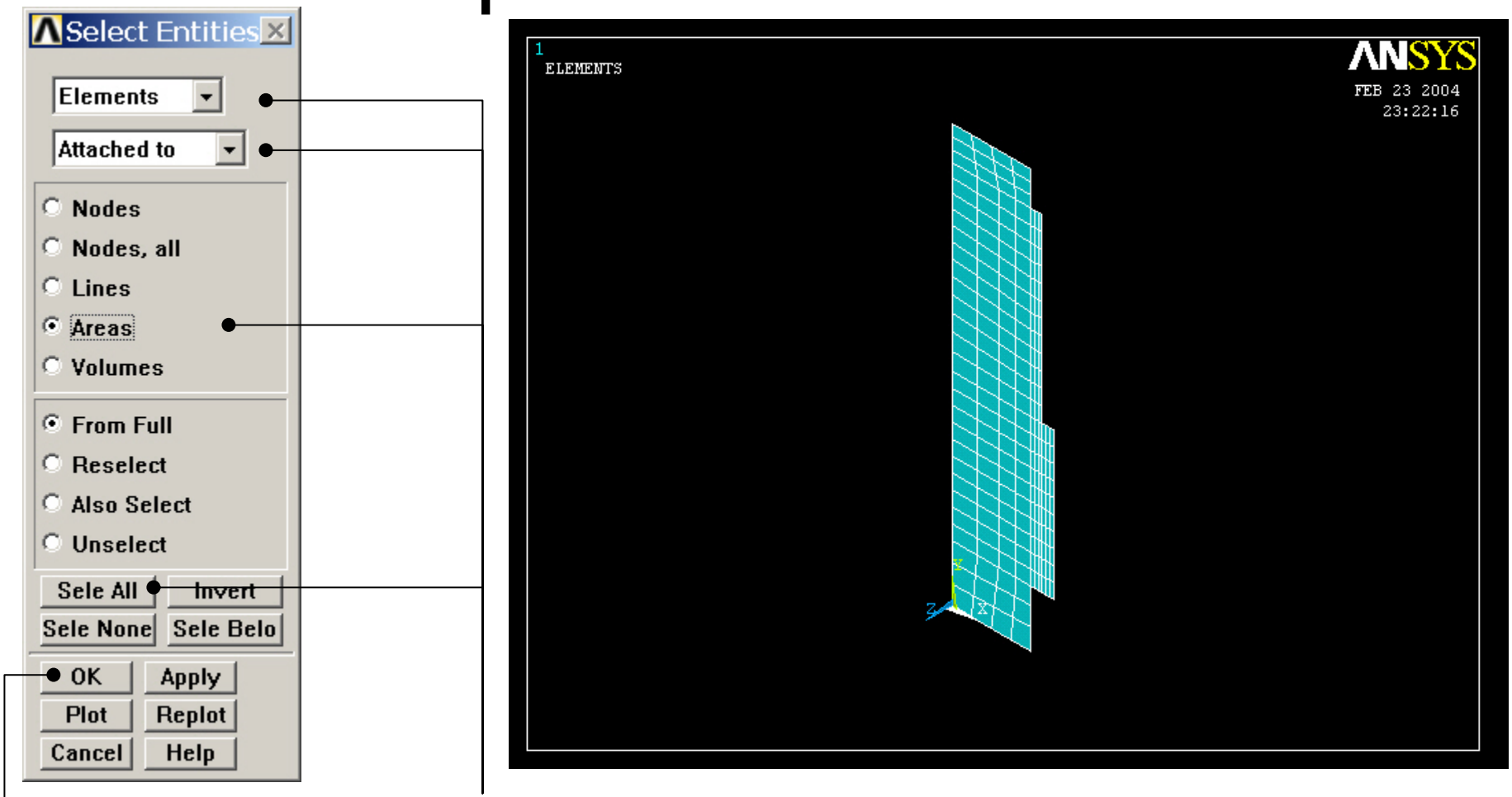


Select indicated areas (use isometric view)

Example – Select entities



Example – Select entities



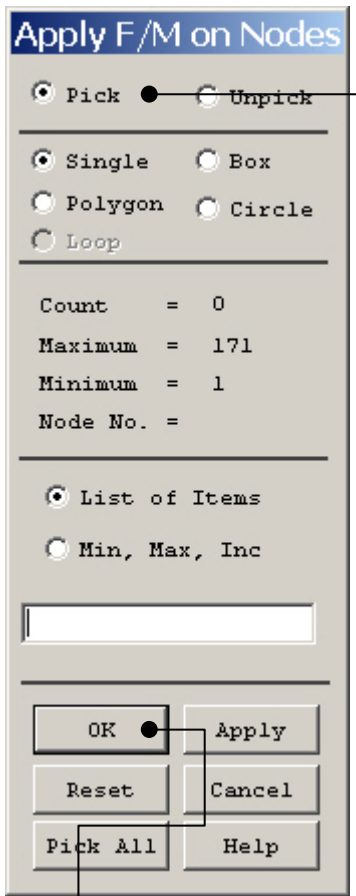
Change to Elements, Attached to, Areas, Sele All

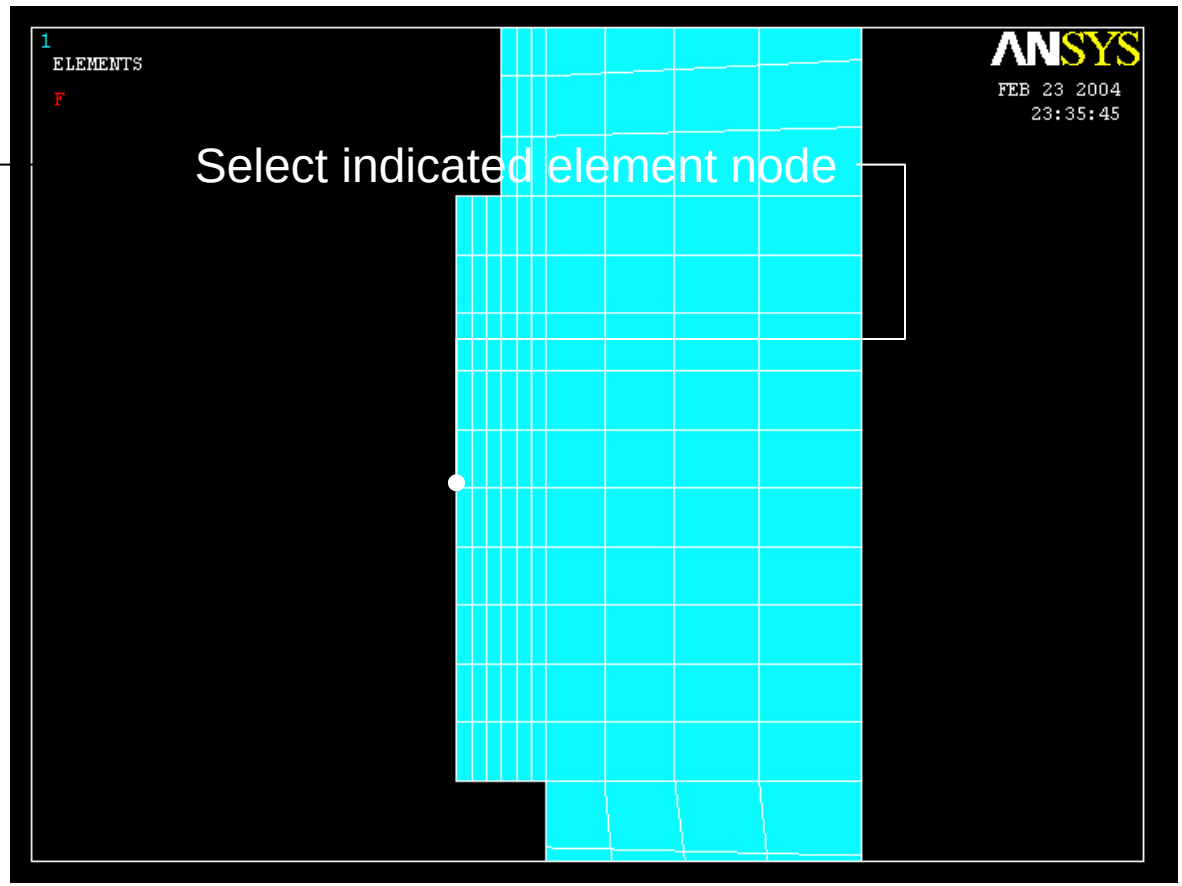
Press OK

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



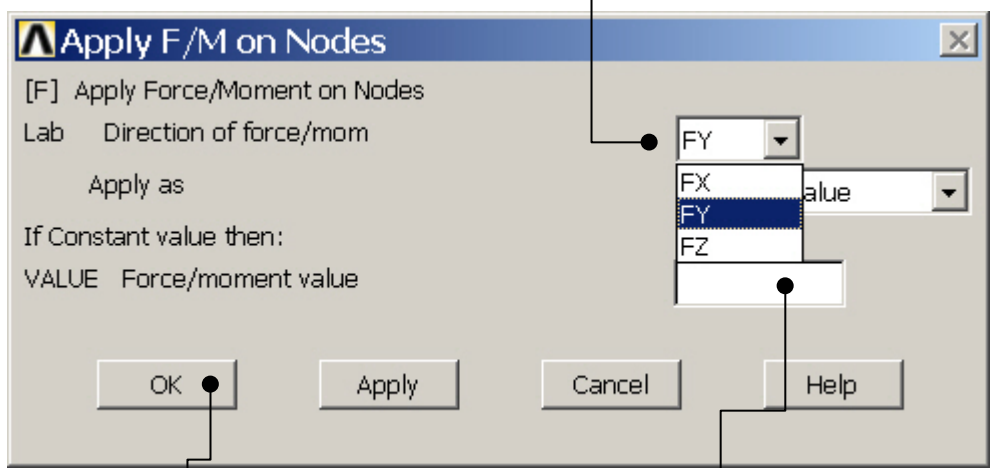


Press OK

Example – Define Loads

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

Change to FZ



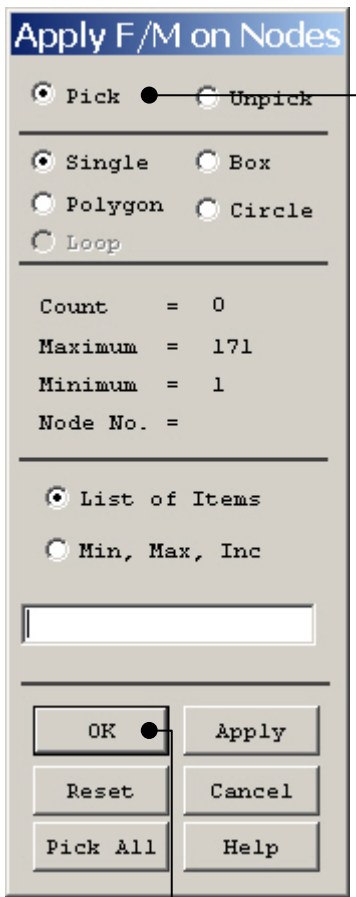
Press OK to finish

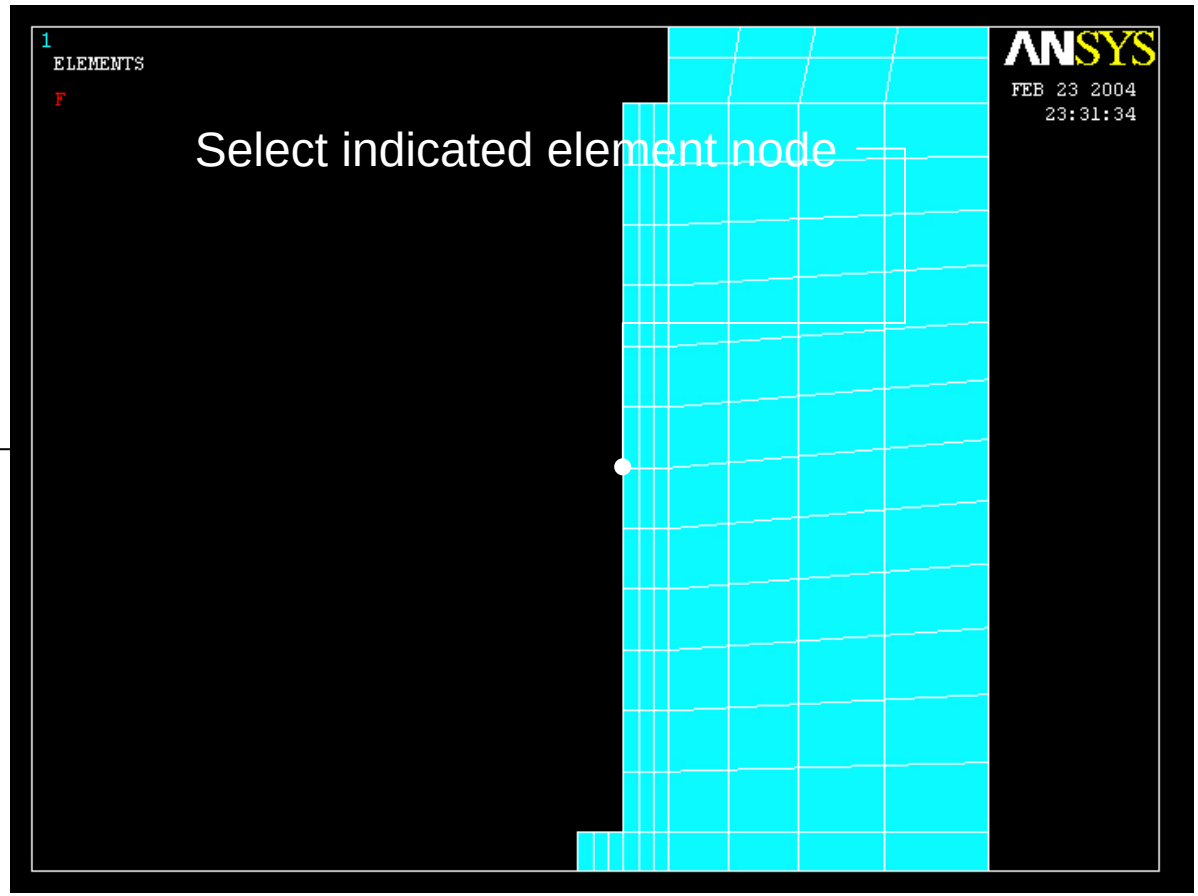
Enter 22.2

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



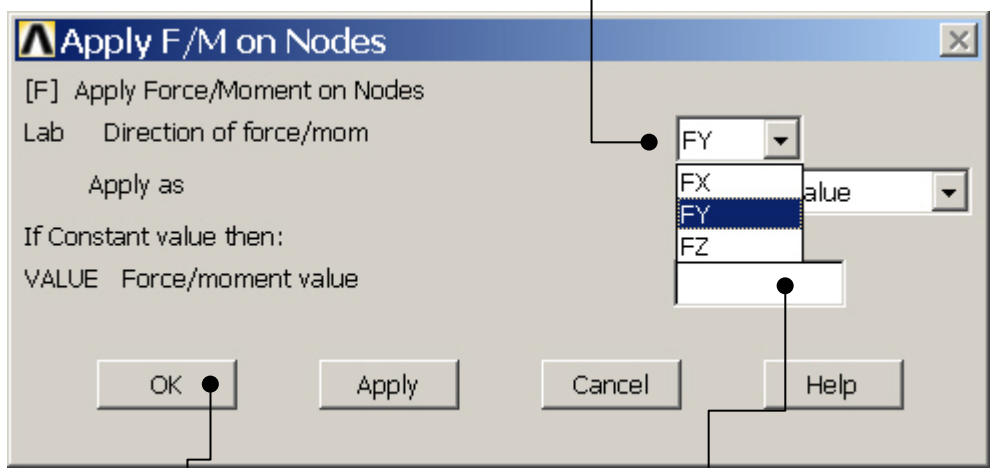


Press OK

Example – Define Loads

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

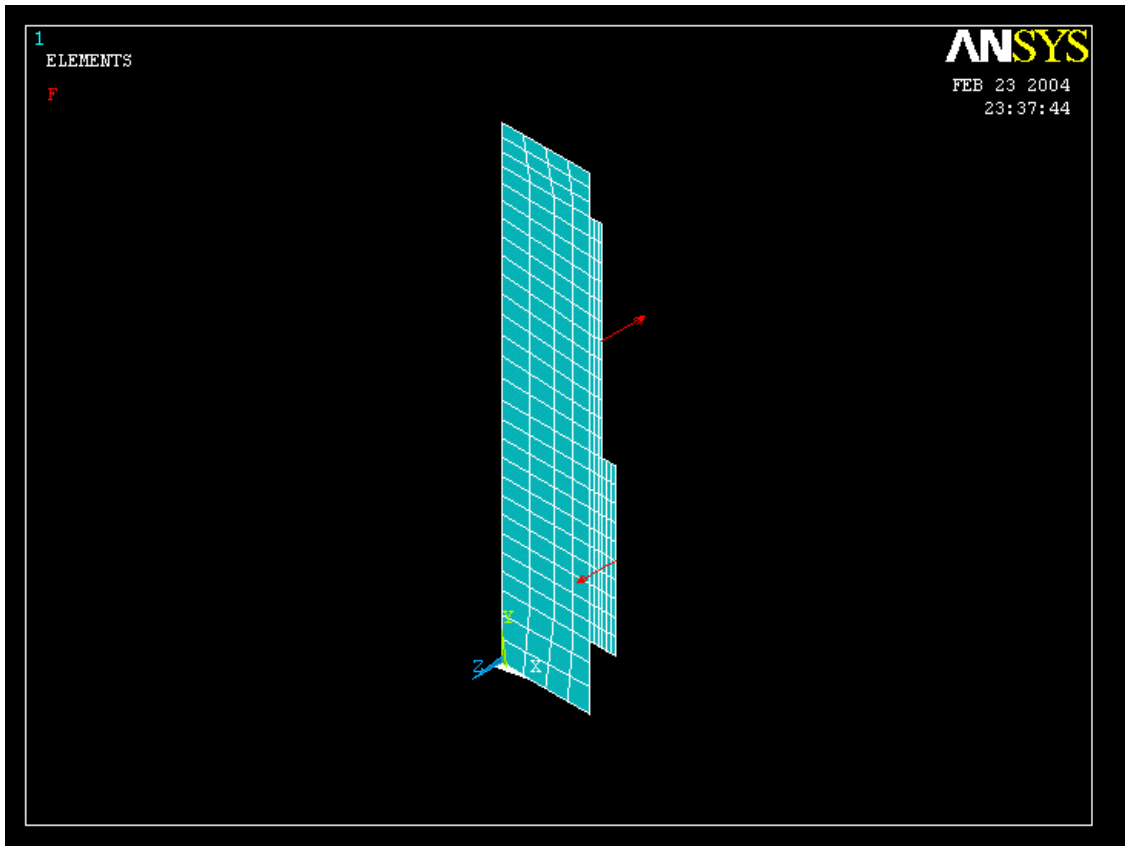
Change to FZ



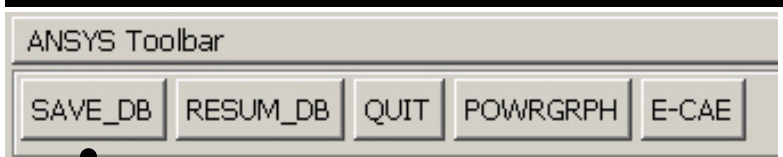
Press OK to finish

Enter -25

Example - Save

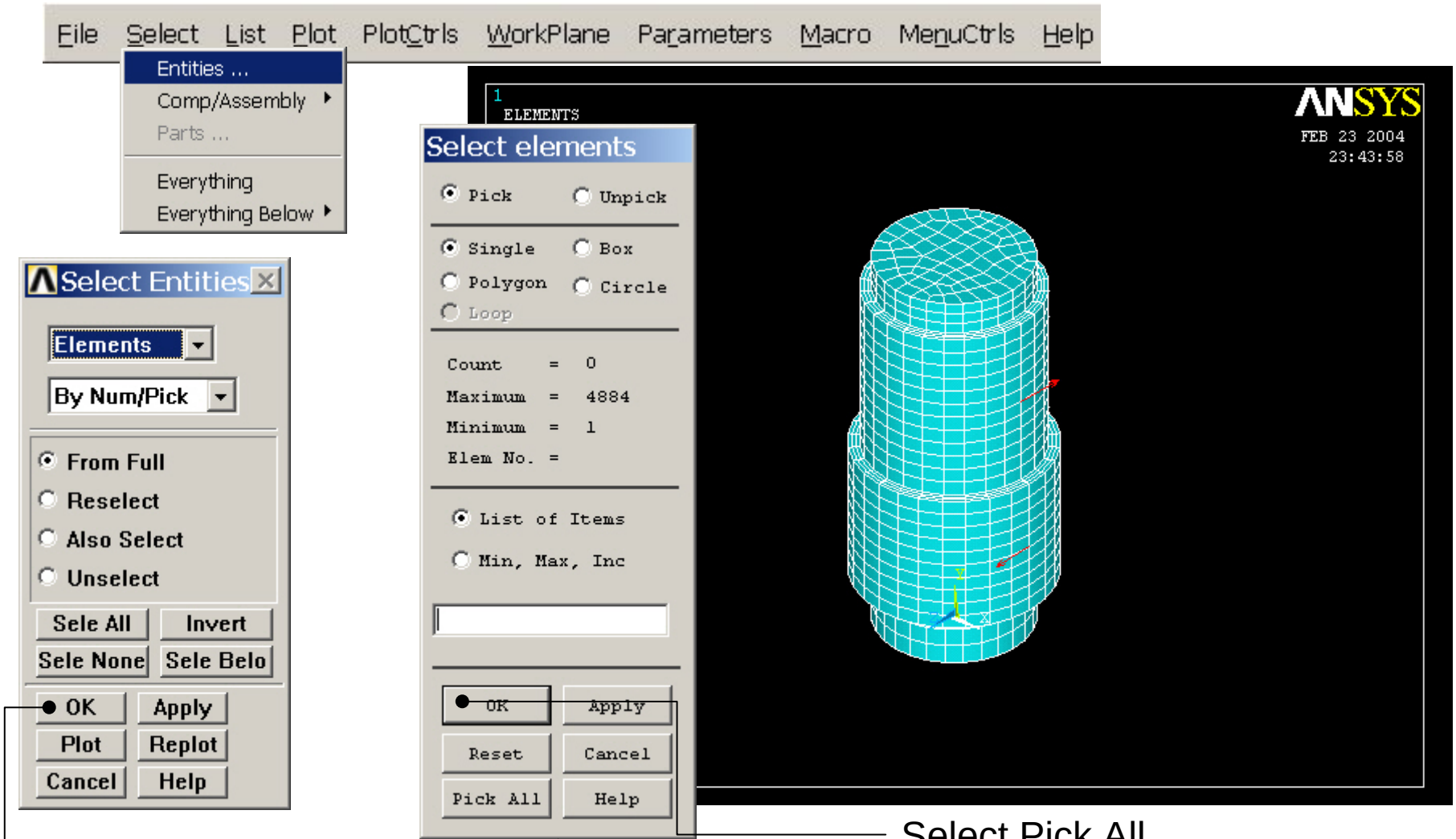


Display of Analysis model



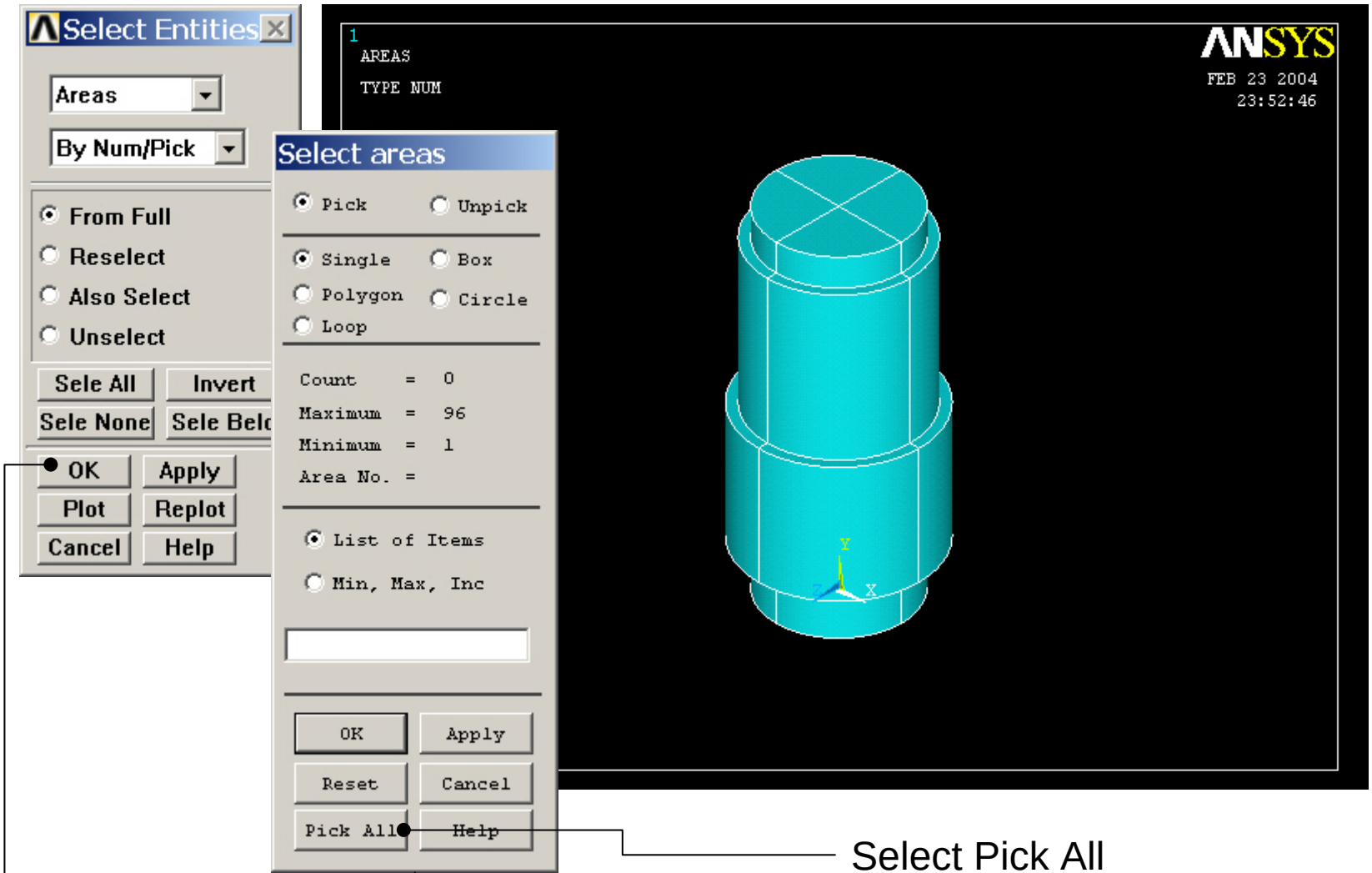
Save the model

Example – Select entities



Select Pick All

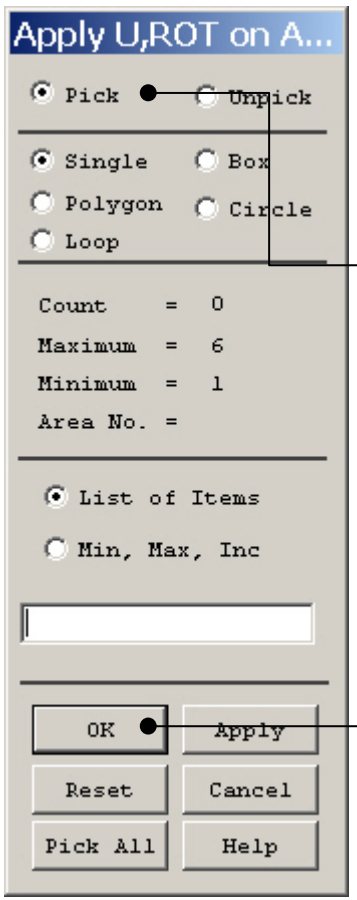
Example – Select entities



Select Pick All

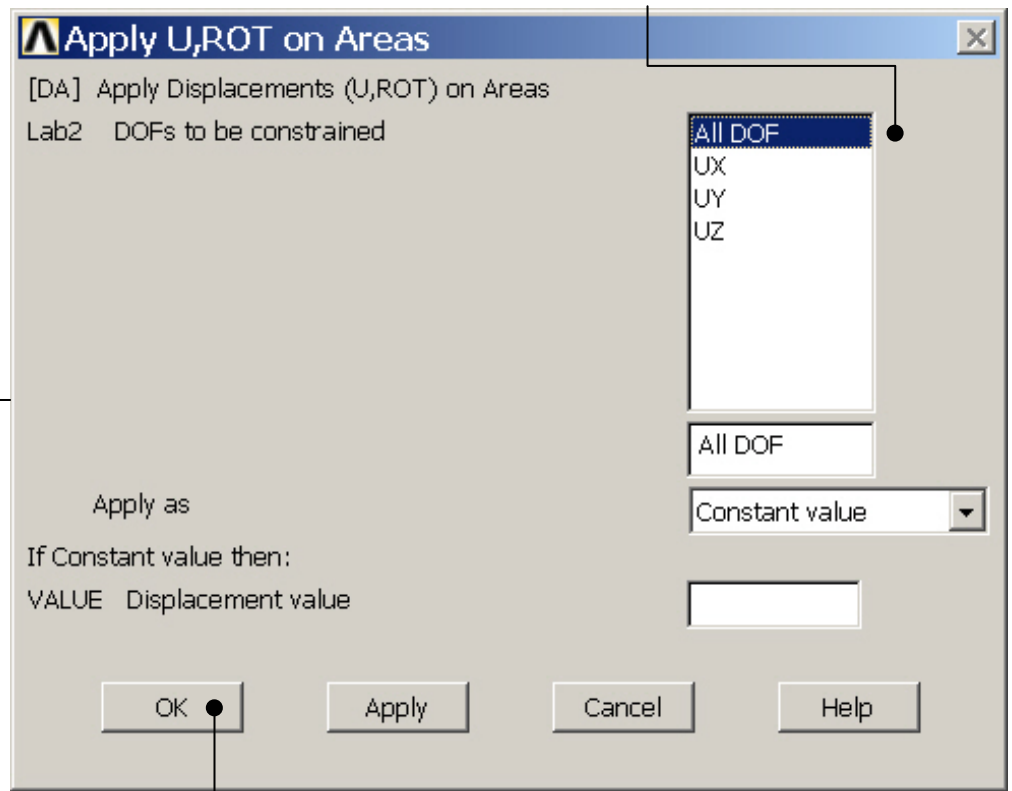
Example – Define Loads

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



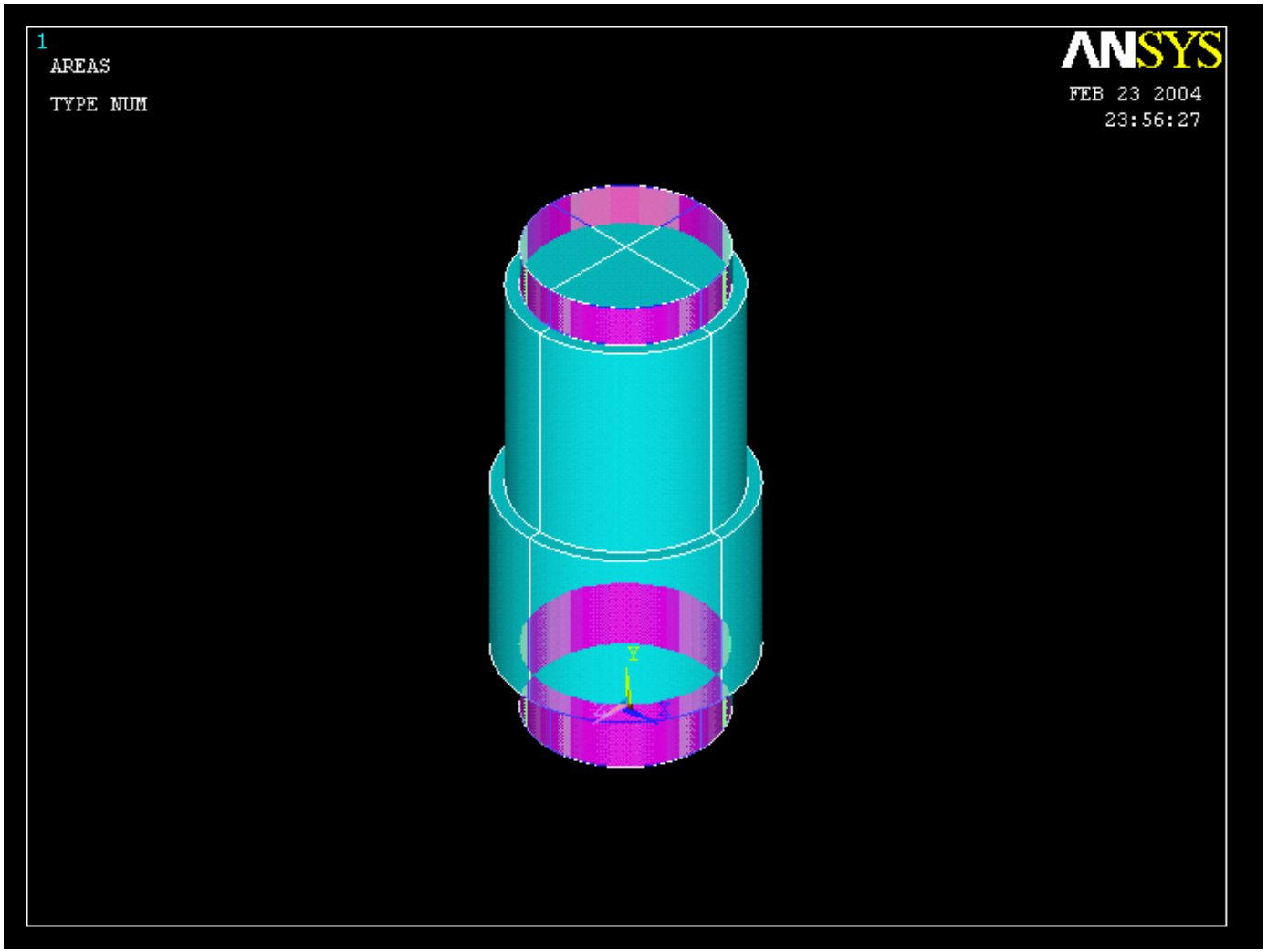
Select Areas as indicated on the next page

Select All DOF to fix/clamp the axle



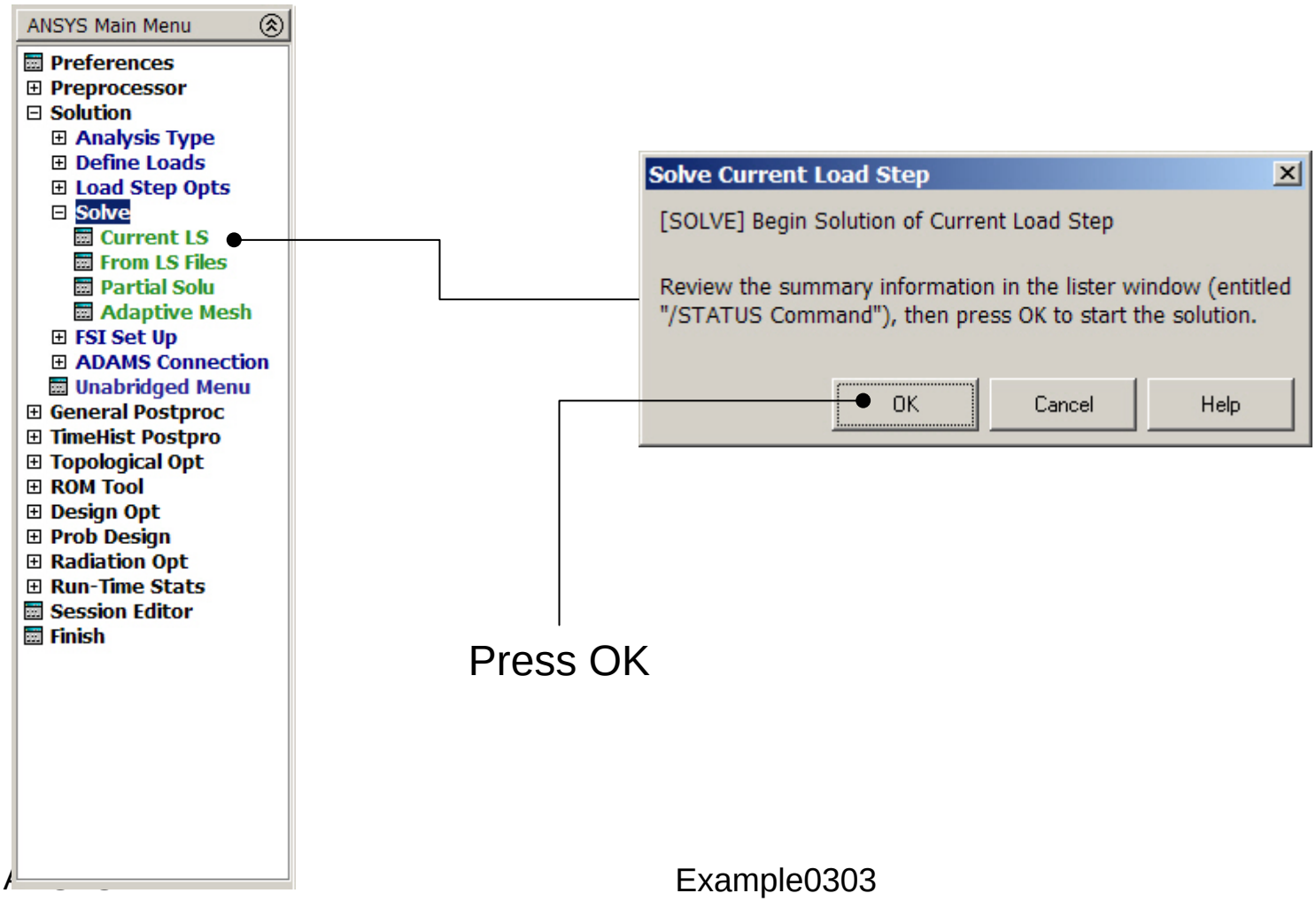
Press OK

Example – Define Loads



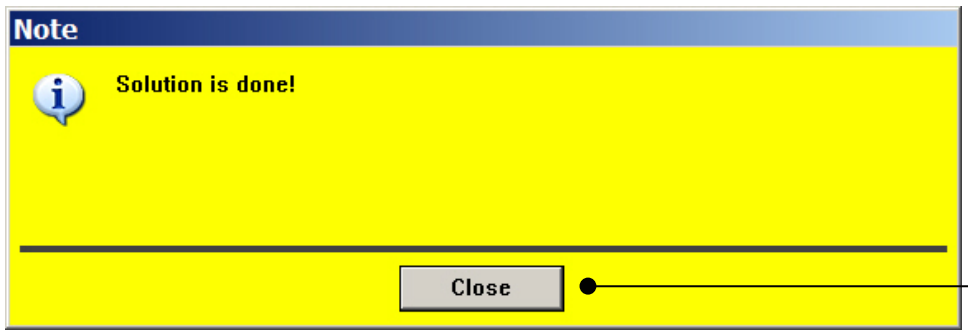
Example - Solve

Solution > Solve > Current LS



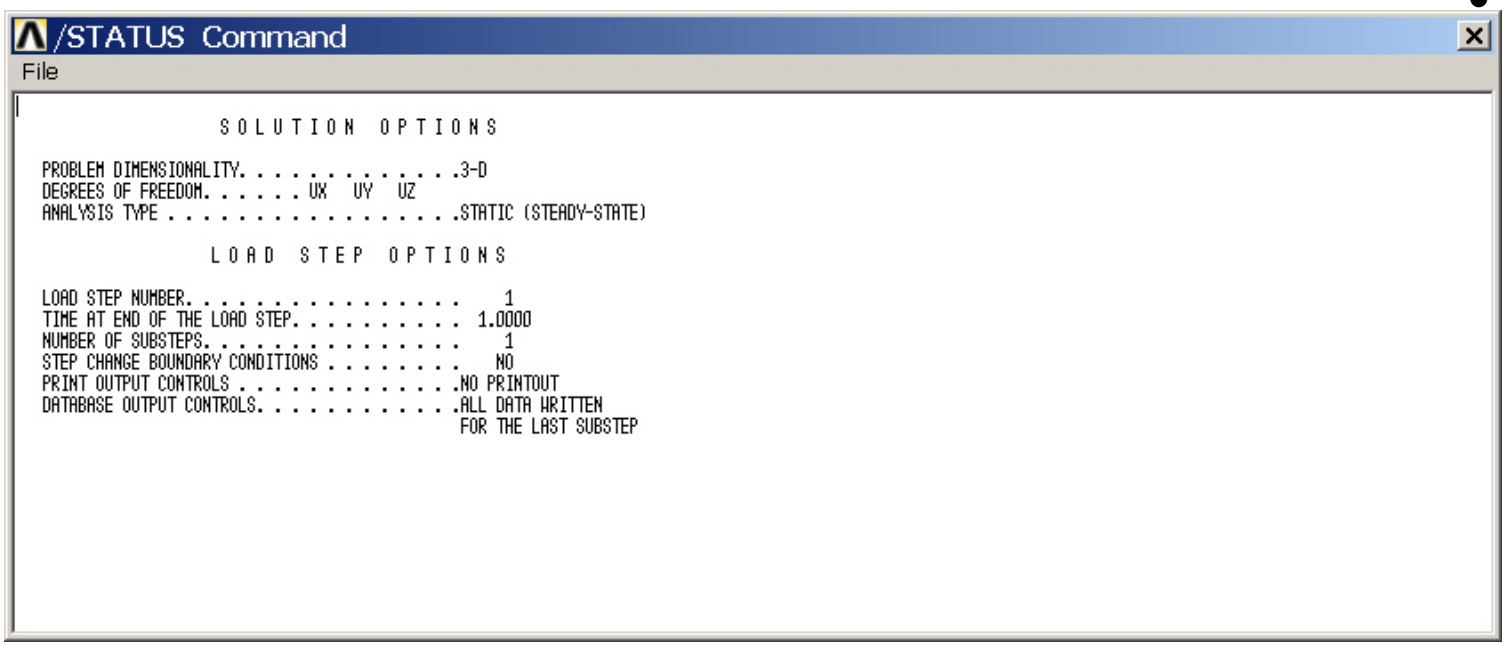
Press OK

Example - Solve

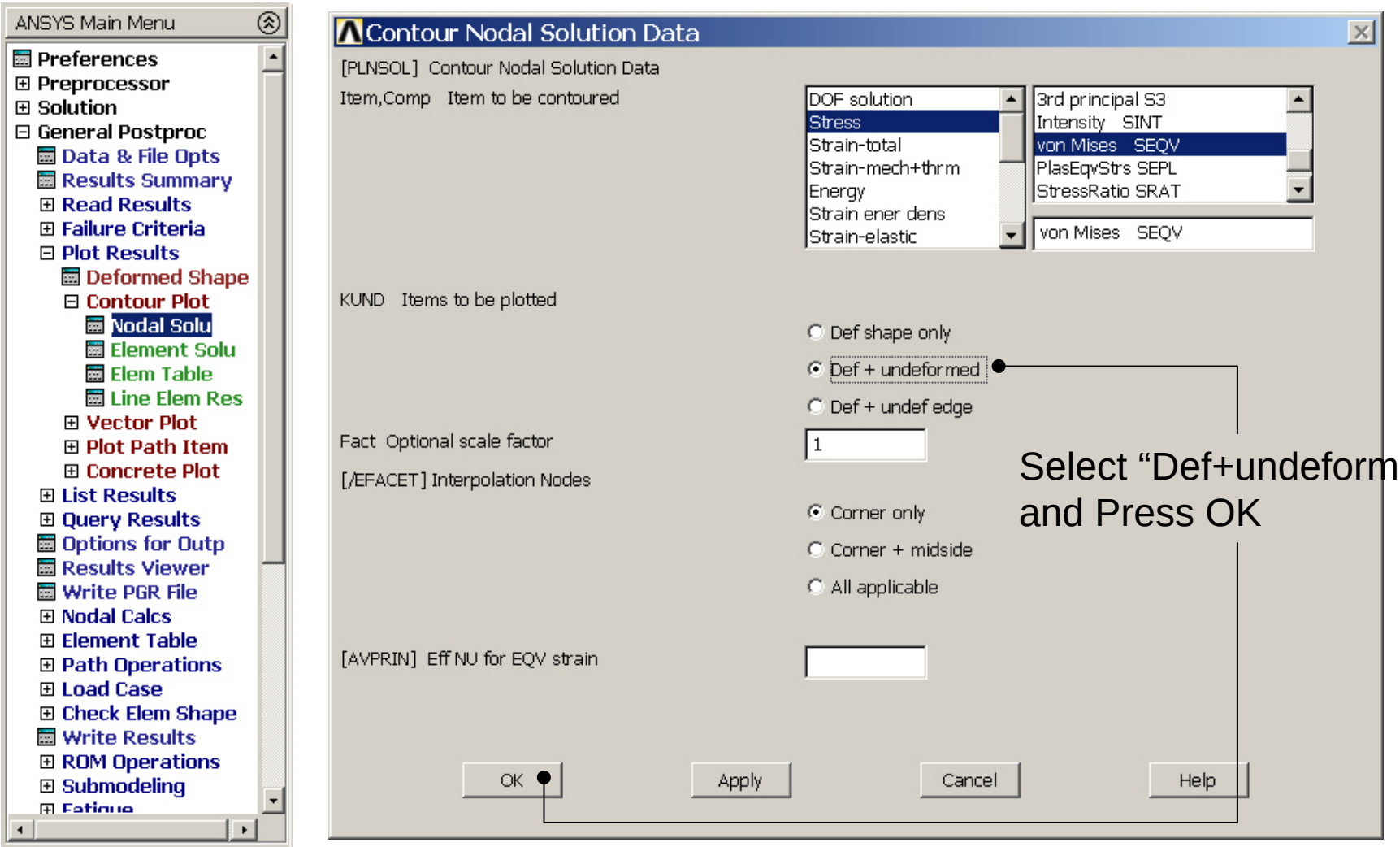


Press Close

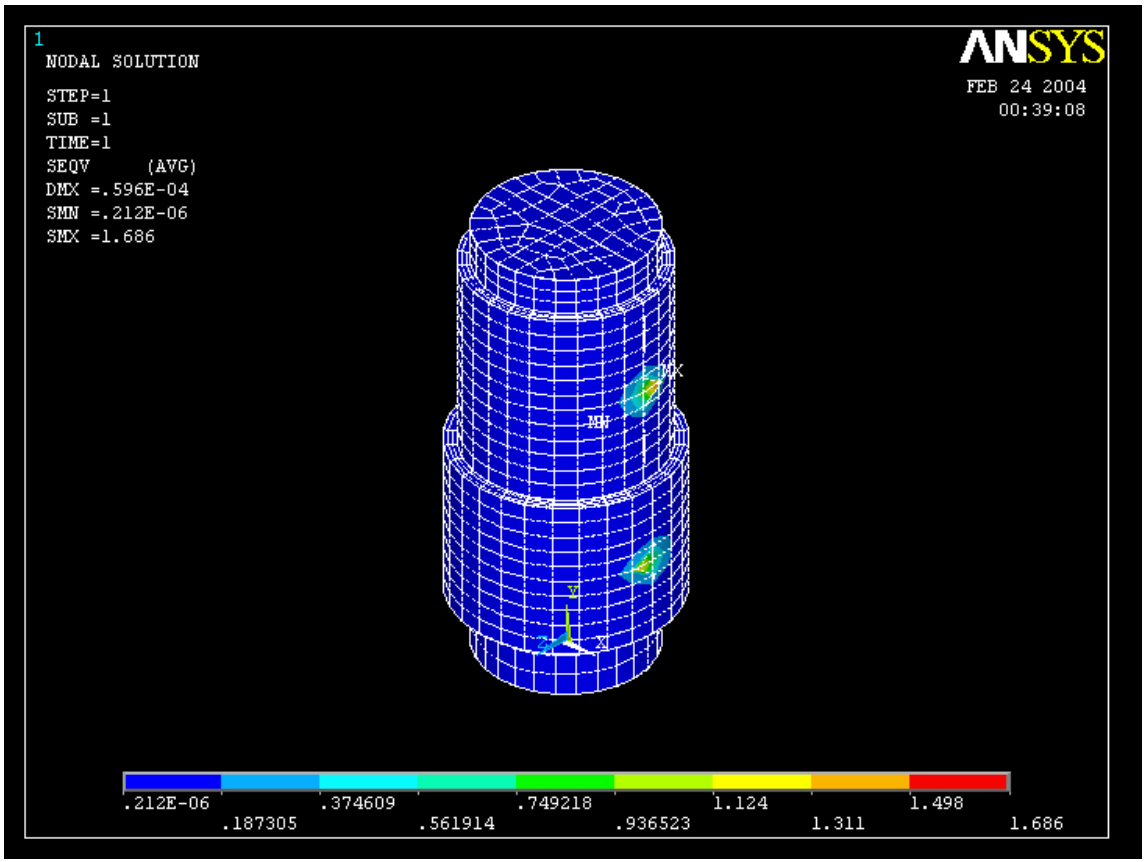
Press here
to Close



Example - PostProcessing



Example - PostProcessing



Read Maximum displacement: DMX