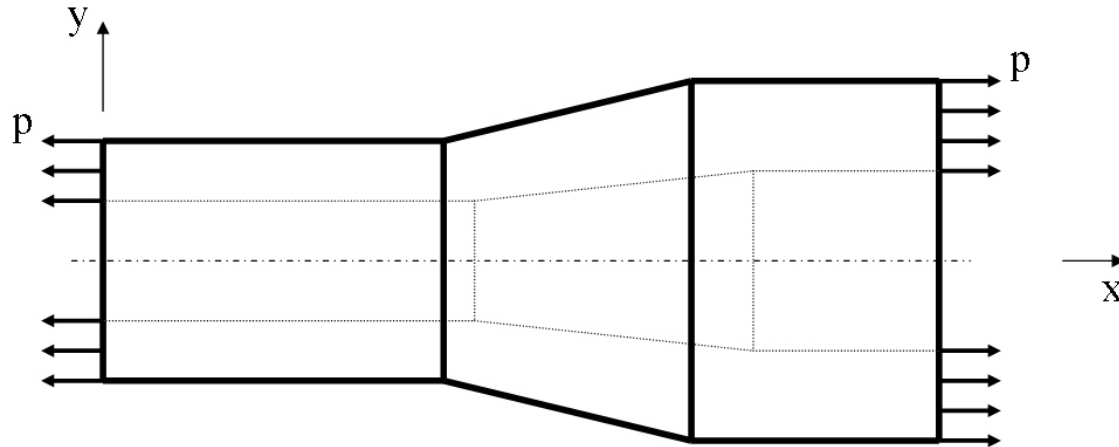


Course in ANSYS

Example0304

Example – Drill pipe 3D



Objective:

Display the stress distribution

Tasks:

How should this be modelled?

Build an axi-symmetric model

Topics:

Element type, Real constants, modeling,

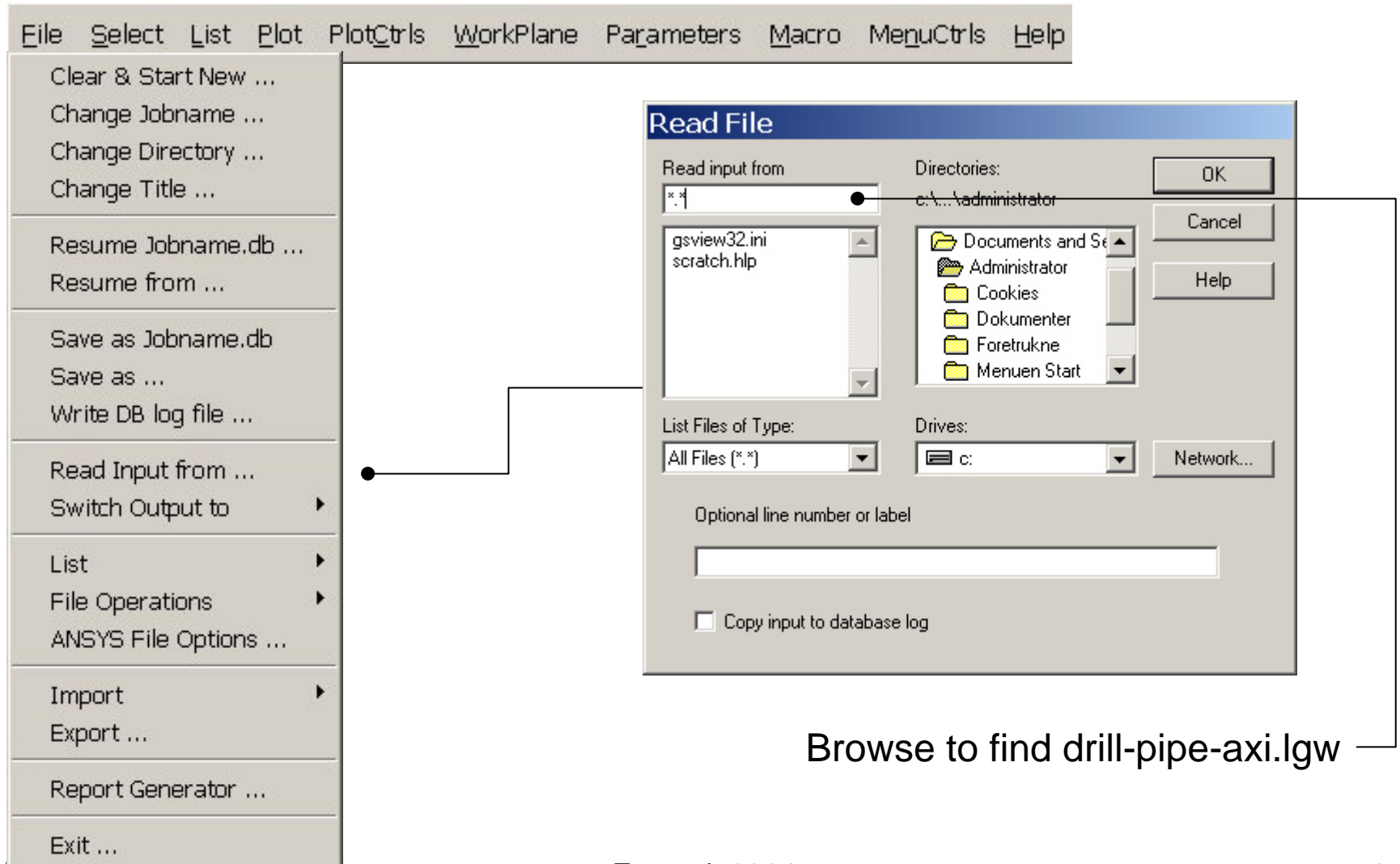
Plot results, output graphics

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

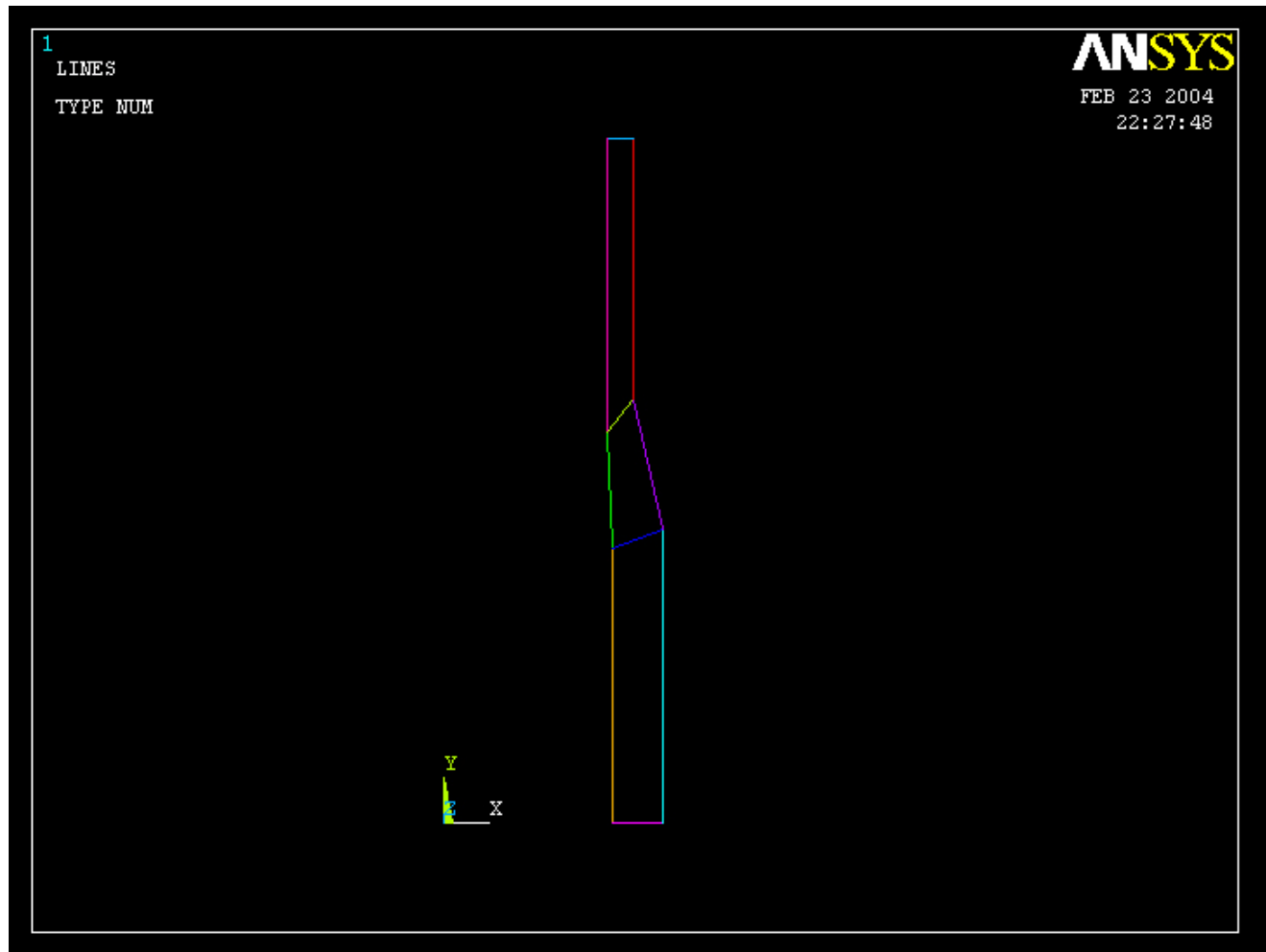
$$\nu = 0.3$$

$$p = 100 \text{ N/mm}$$

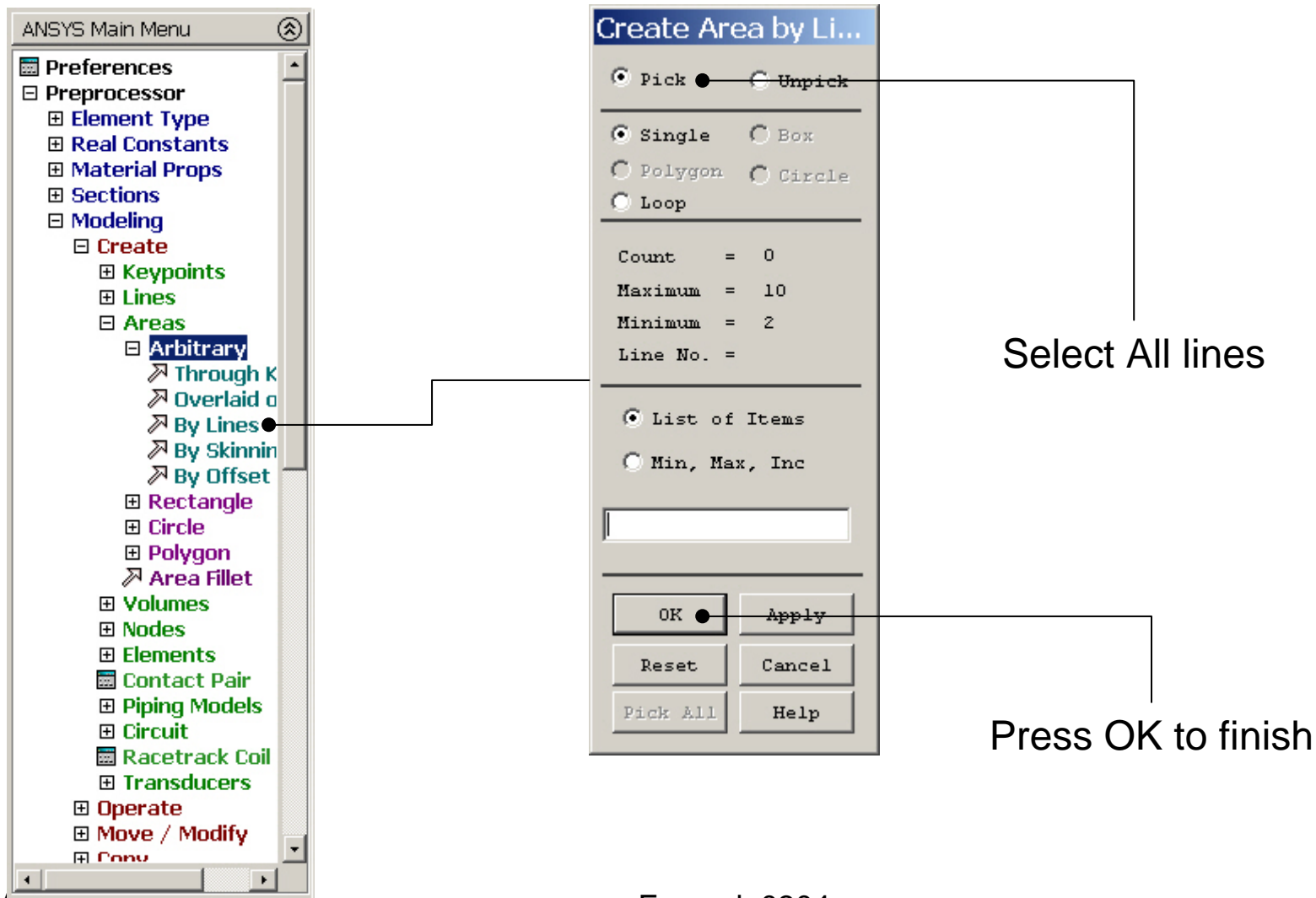
Example – Read Input from



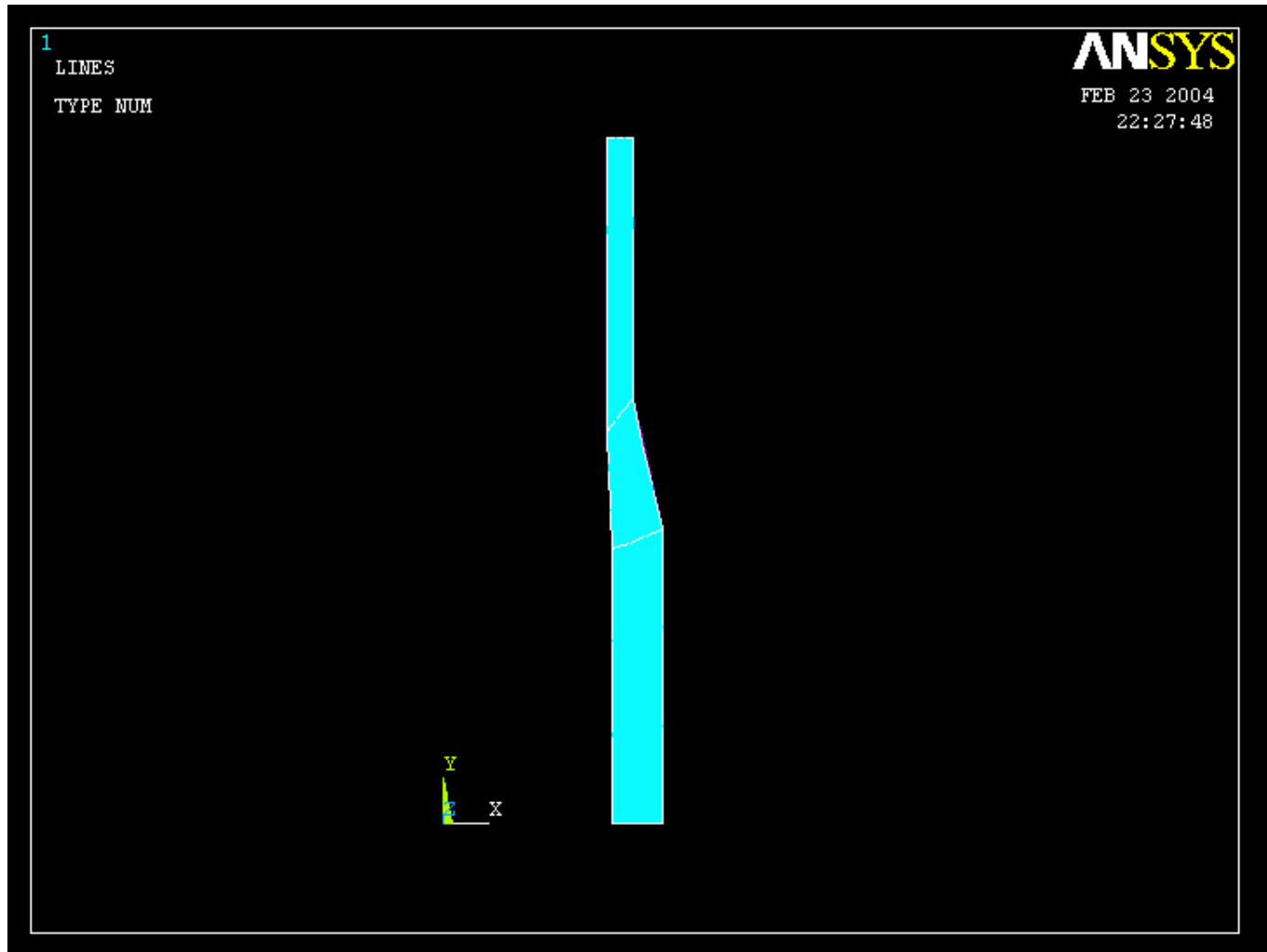
Example – Read Input from



Example – Create Areas by line

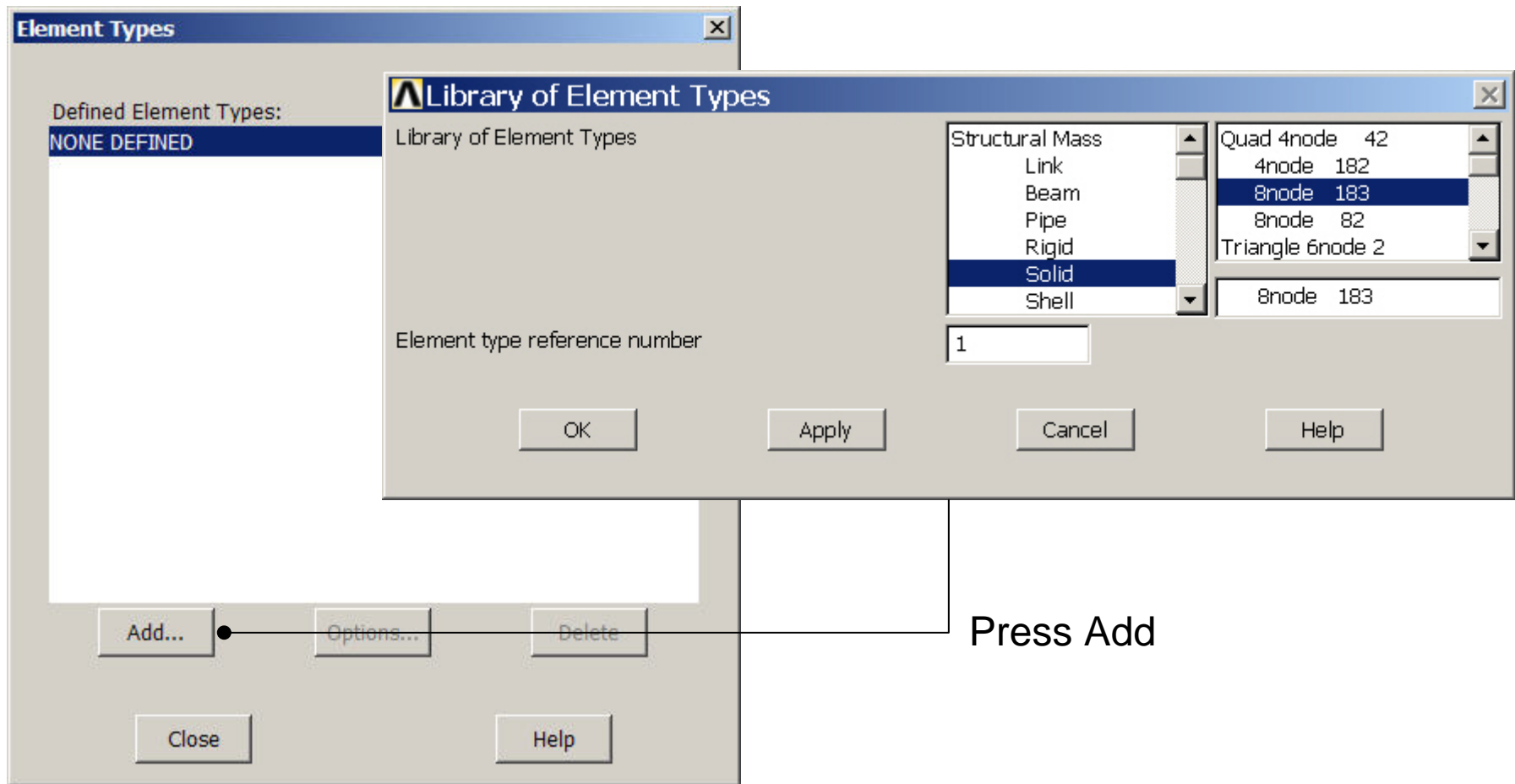


Example – Create Areas by line



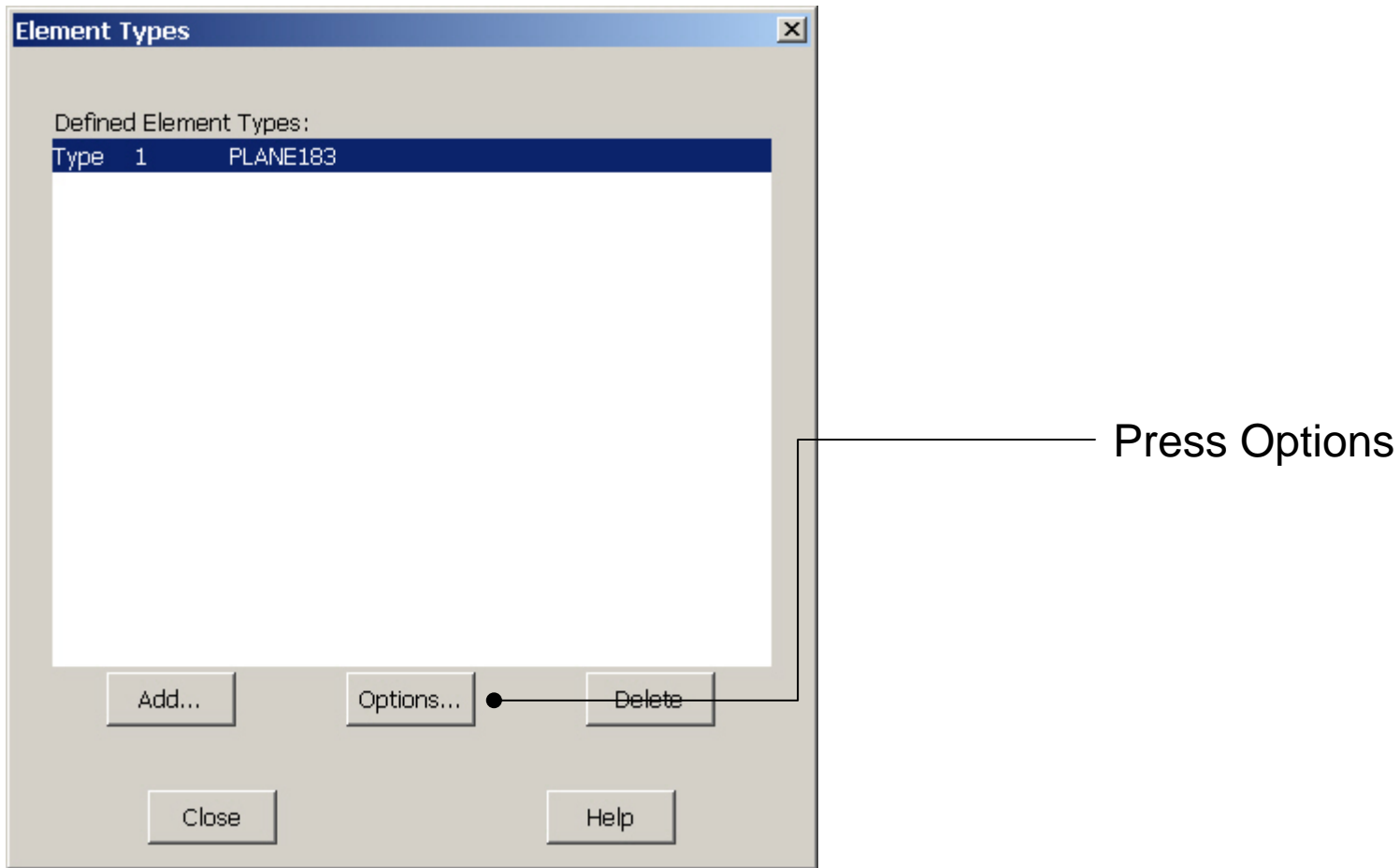
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



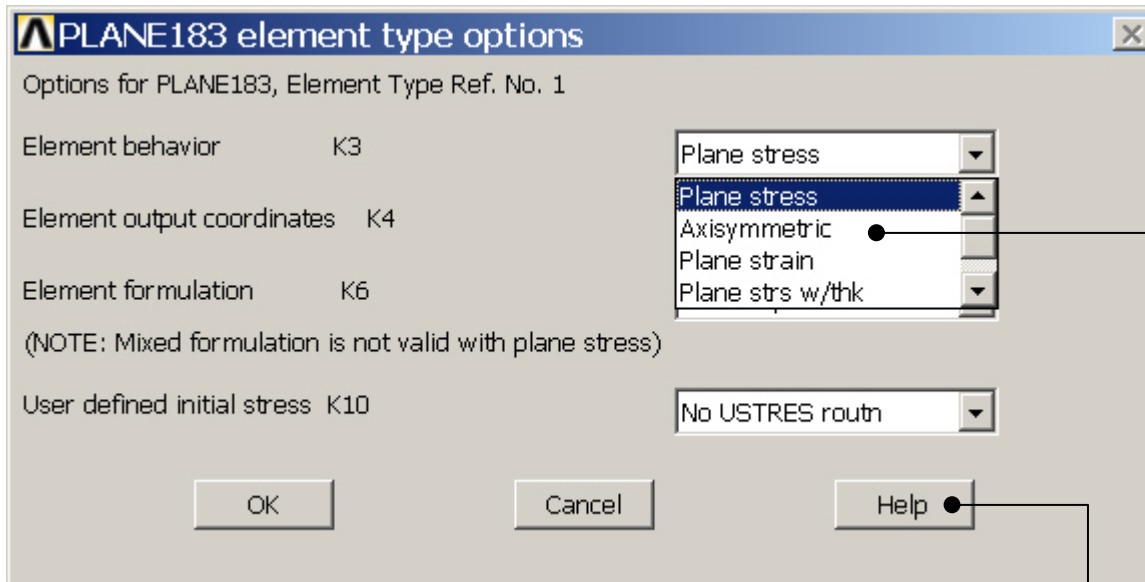
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Select Axisymmetric

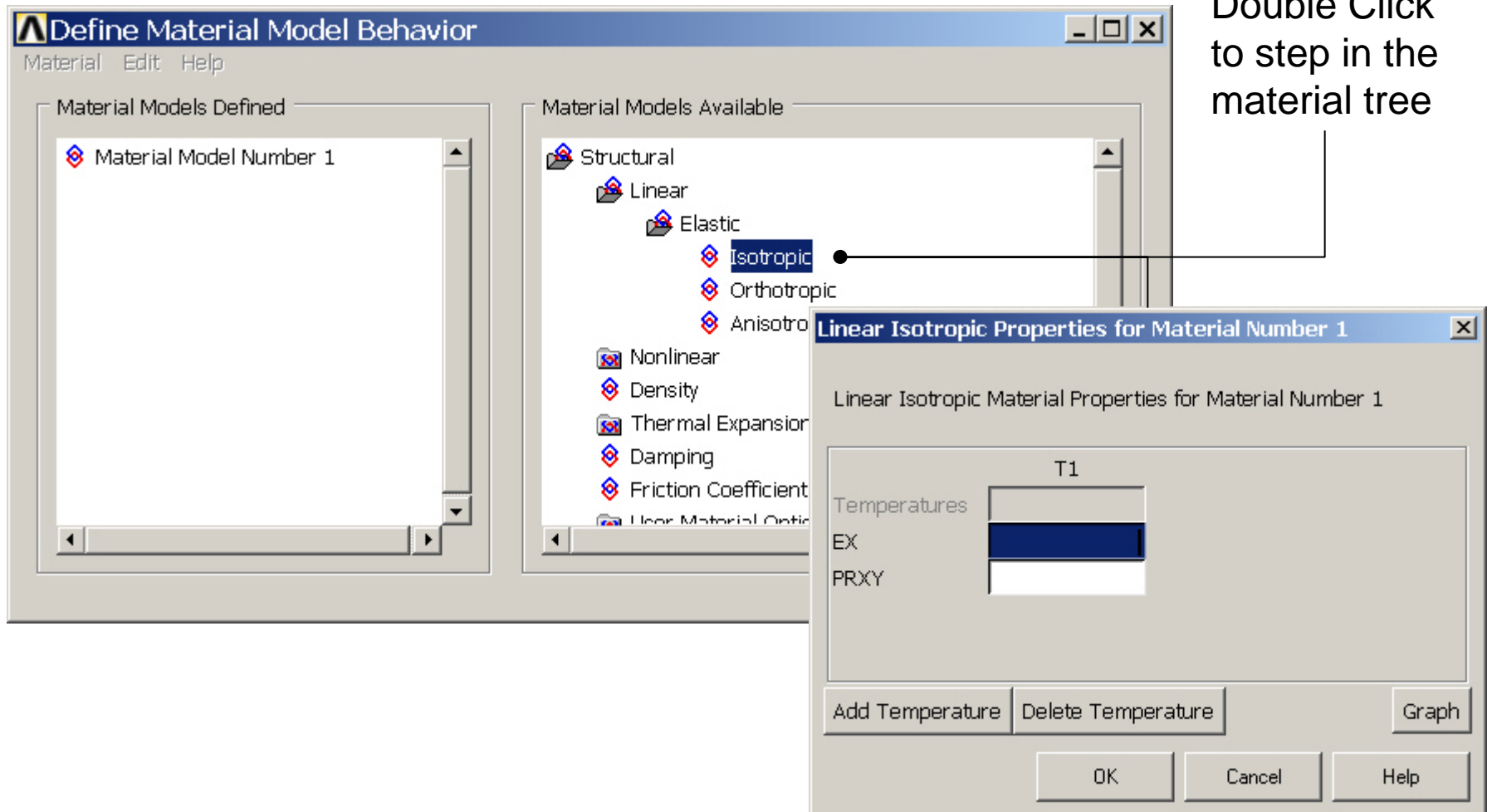
Press Help to learn more about the element.

Example – Real Constants

No Real Constants are necessary for axi-symmetric models!

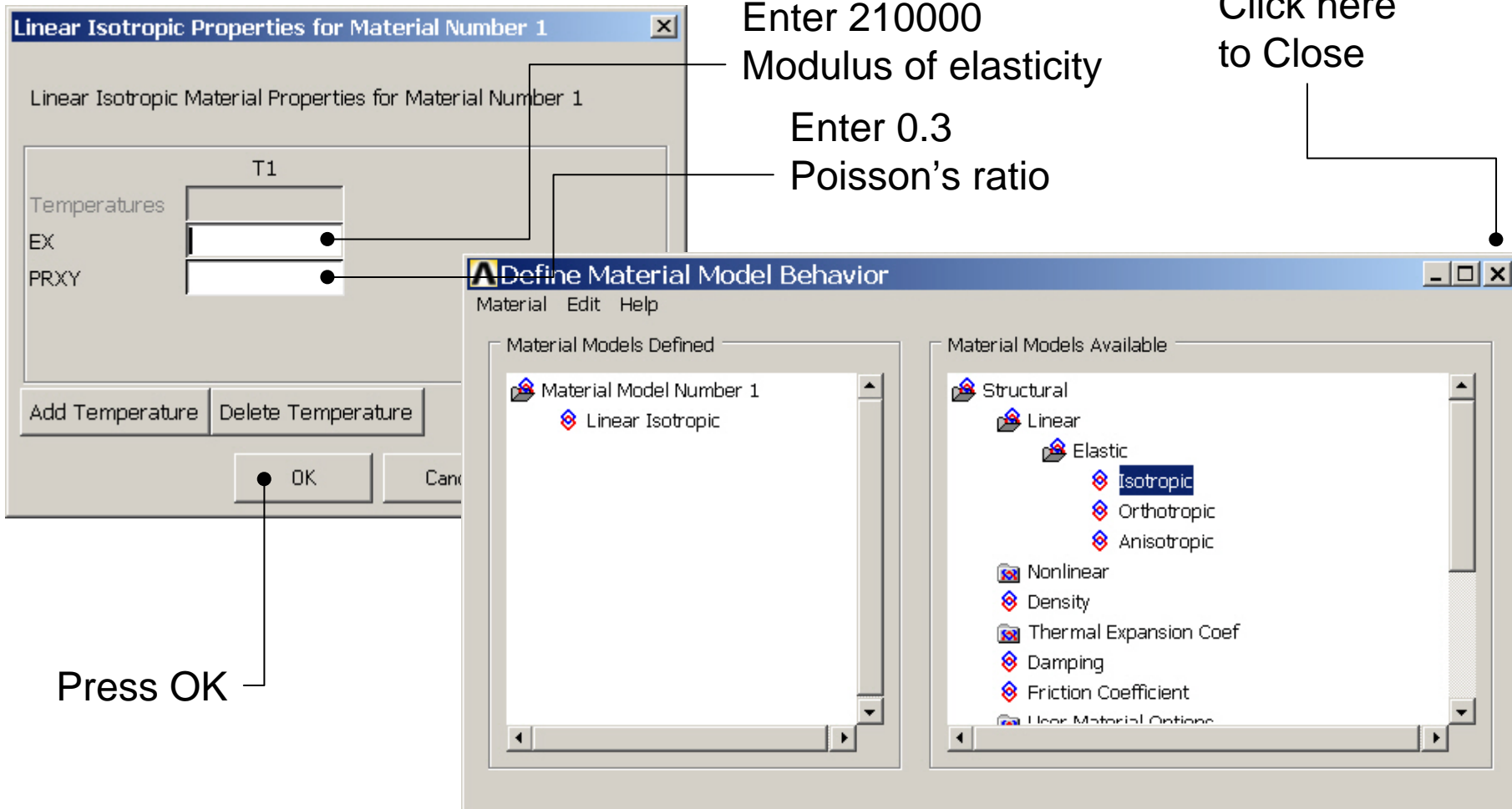
Example - Material Properties

Preprocessor > Material Props > Material Models



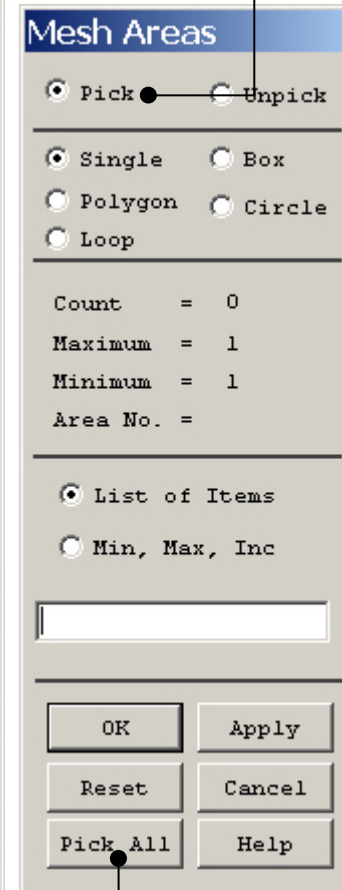
Example - Material Properties

Preprocessor > Material Props > Material Models



Example - Meshing

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

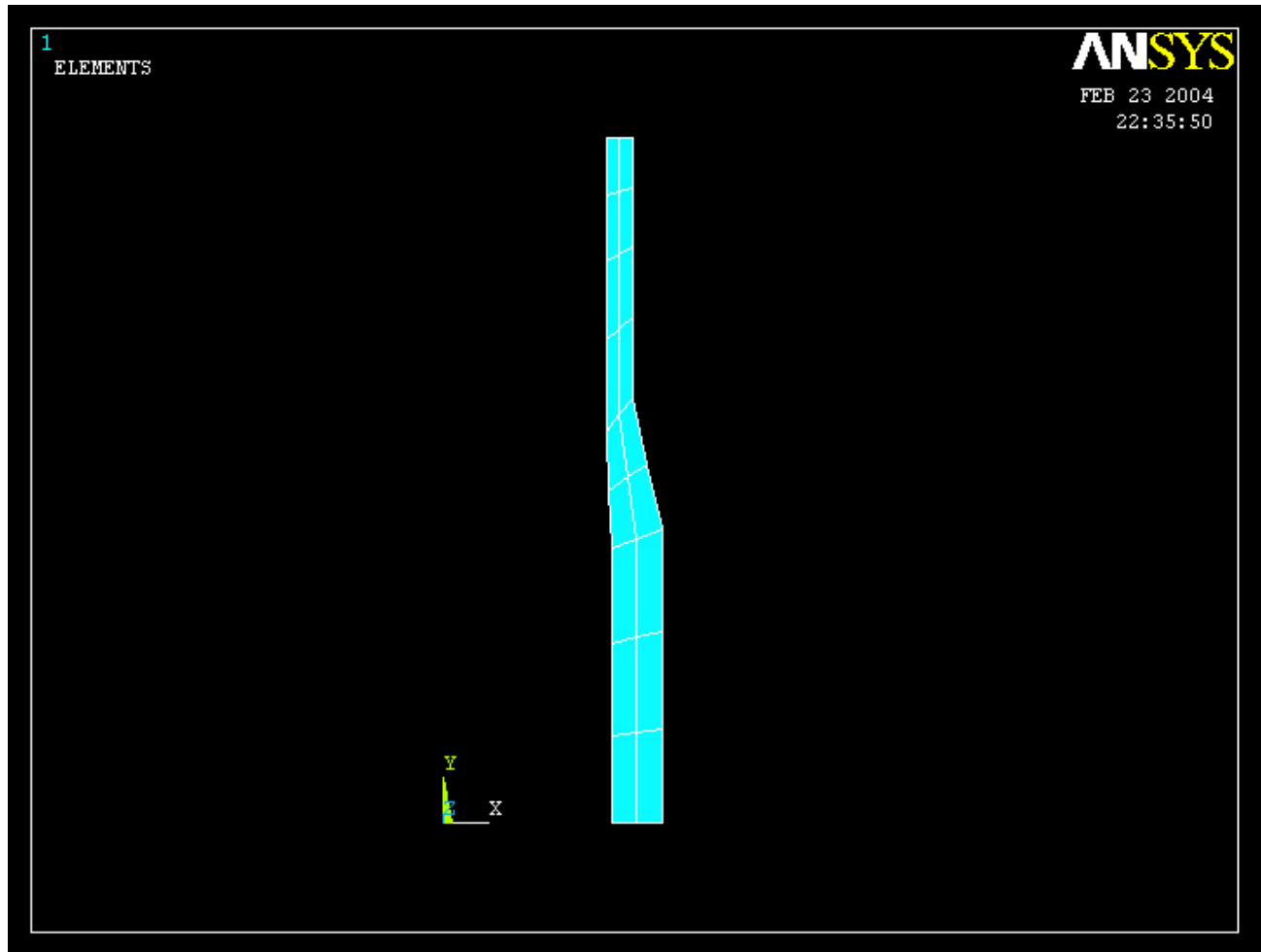


Select individual areas to be meshed

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

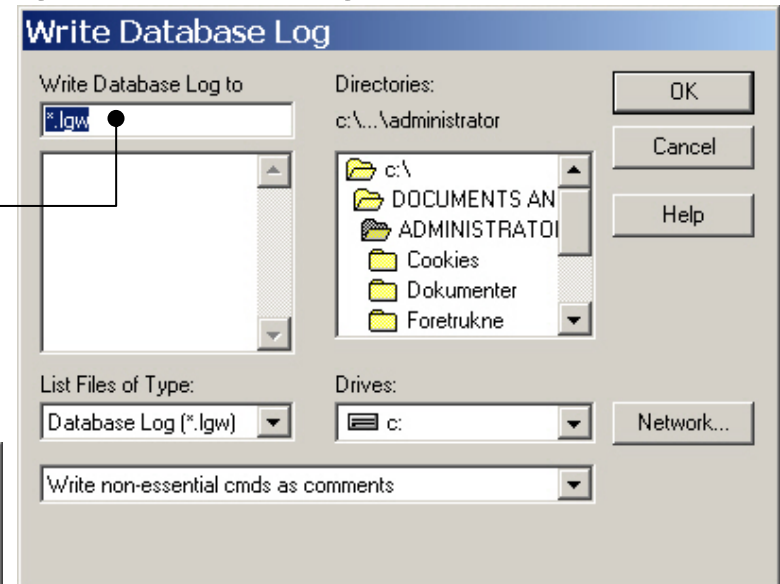
Select all areas defined to be meshed

Example - Meshing

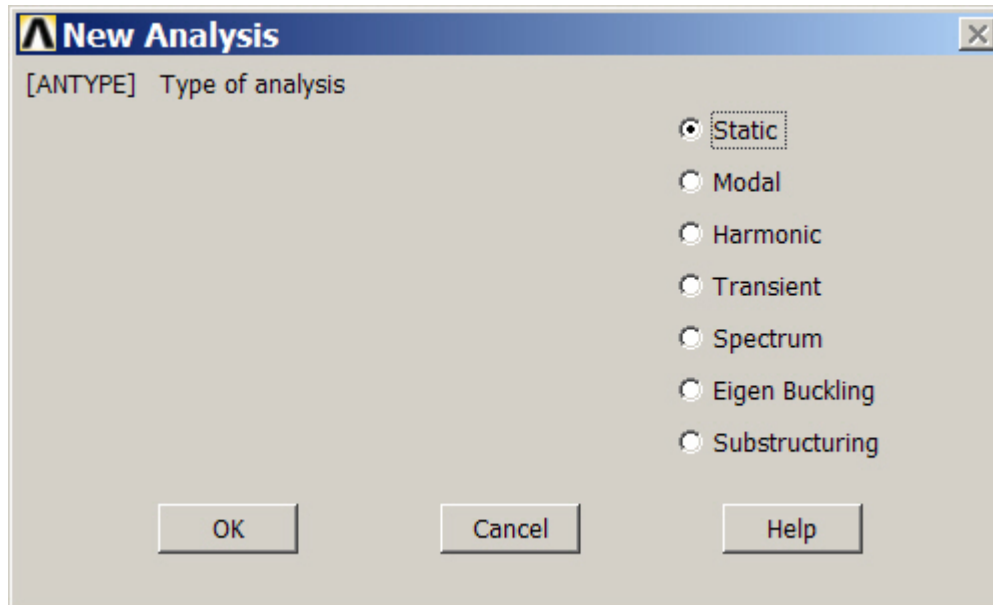


Example – Analysis Type

File > Write DB log file
Enter “example0304.lgw”

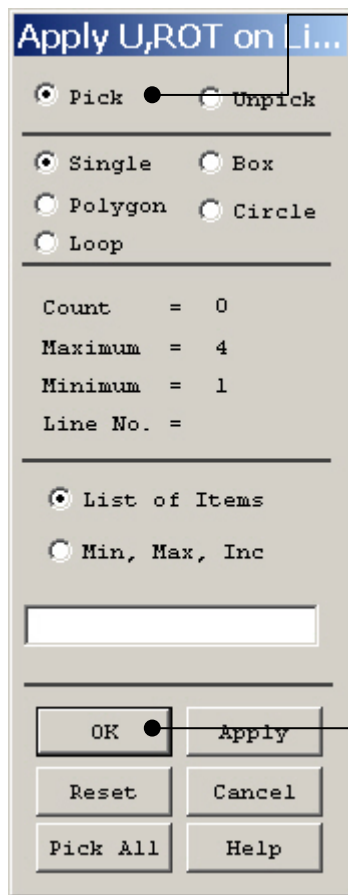


Solution > Analysis Type > New Analysis



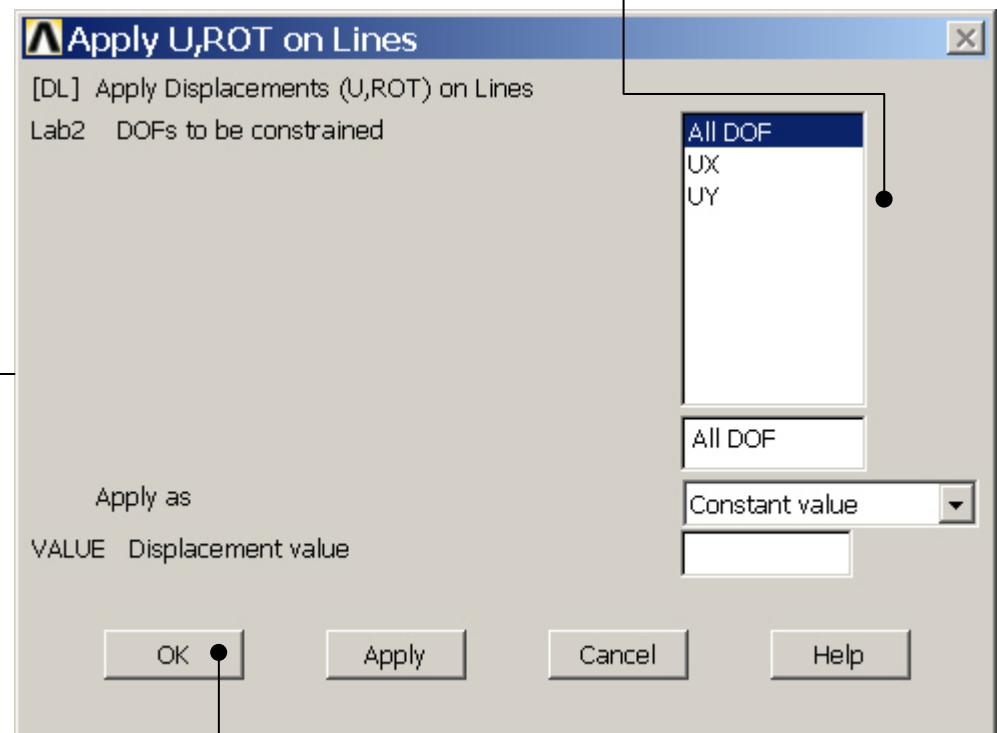
Example – Define Loads

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



Select Line L8

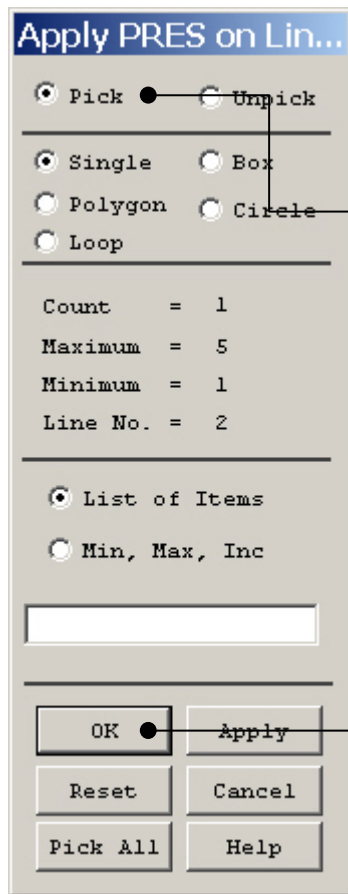
Select UY to fix the pipe in the y-direction



Press OK

Example – Define Loads

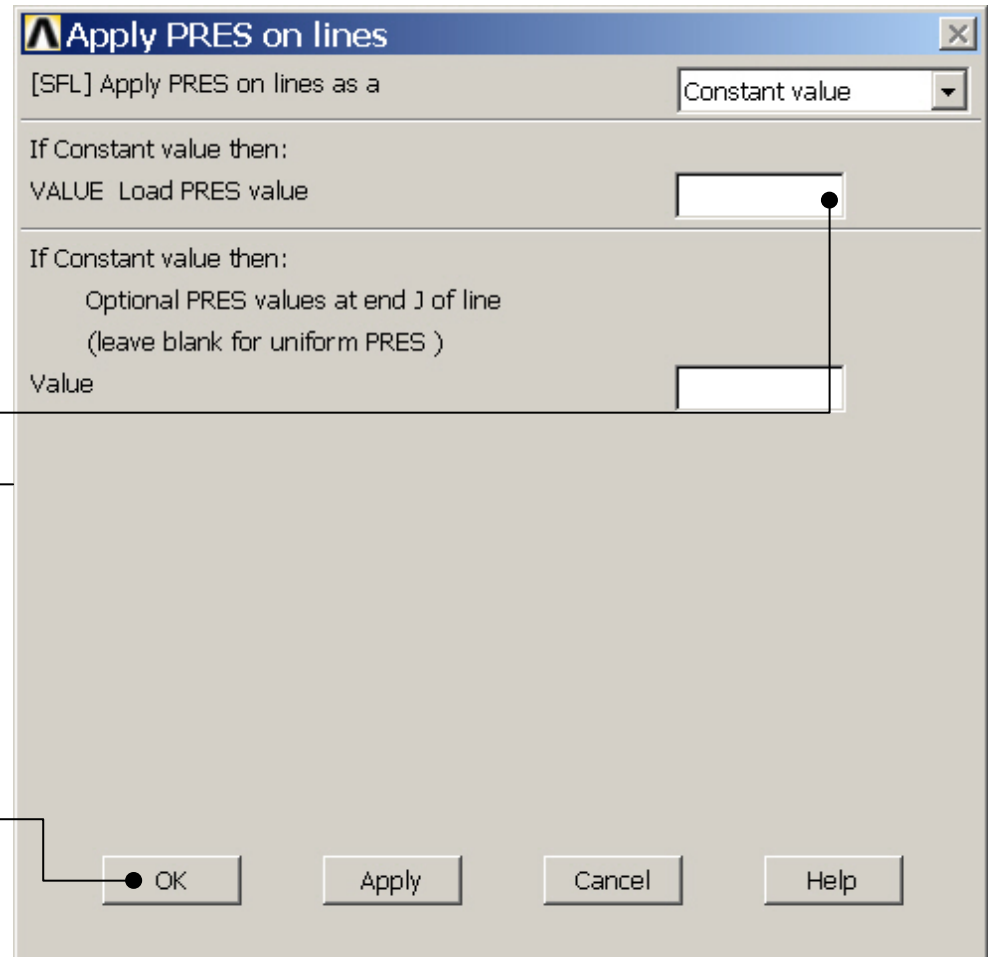
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select line
L4

Enter -100

Press OK
to finish



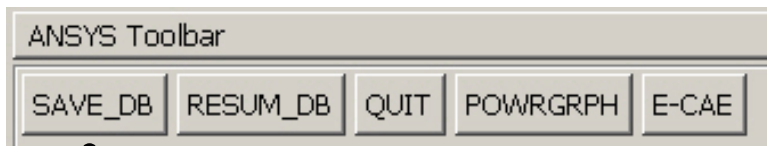
Note: Pressure acts normal and
inward to a surface
ANSYS
Computational Mechanics, AAU, Esbjerg

Example0304

Example - Save



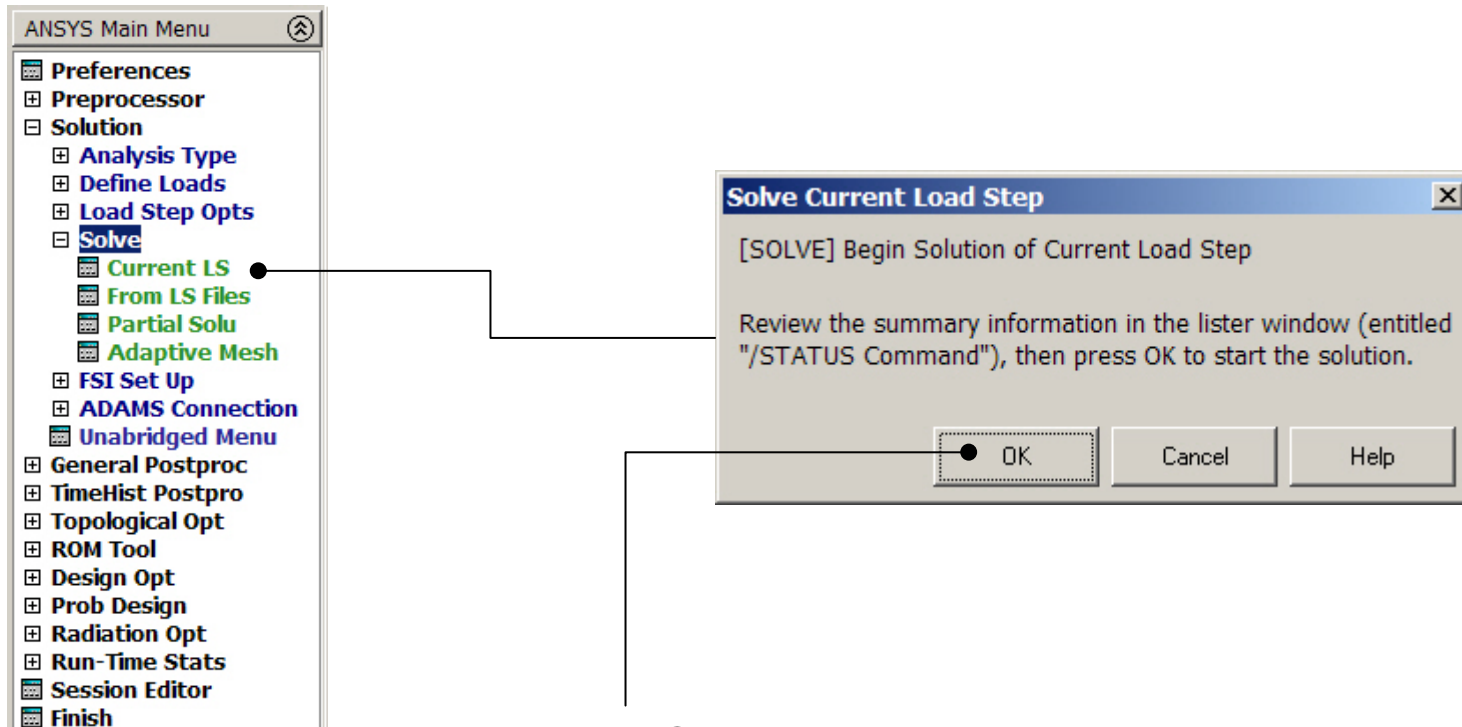
Display of Analysis model



Save the model

Example - Solve

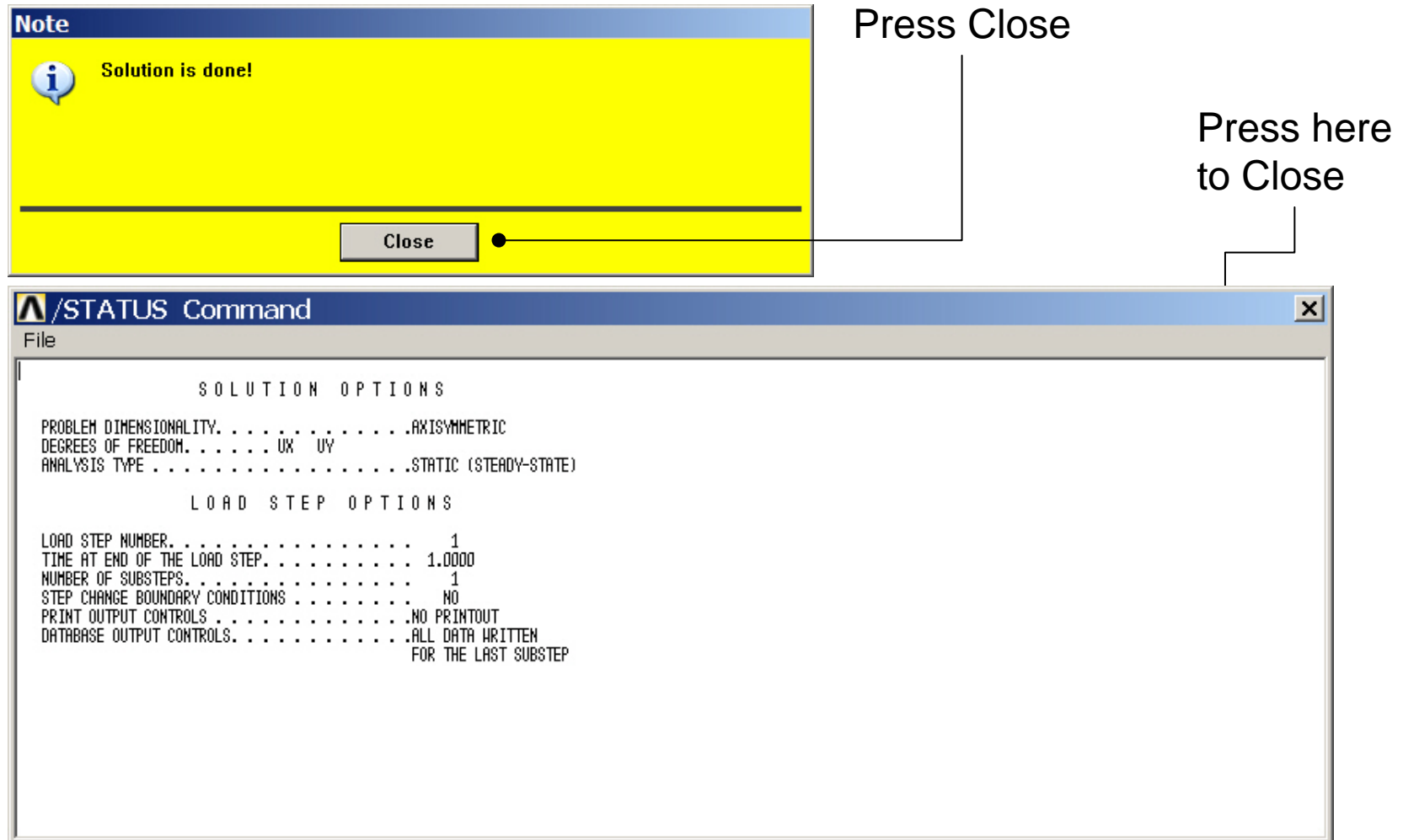
Solution > Solve > Current LS



Press OK

Example0304

Example - Solve



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 the following items: 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, and Fatigue. The Contour Nodal Solution Data dialog box has the following fields and options: [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, and Help. A text box with the text "Select 'Def+undeformed' and Press OK" has 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

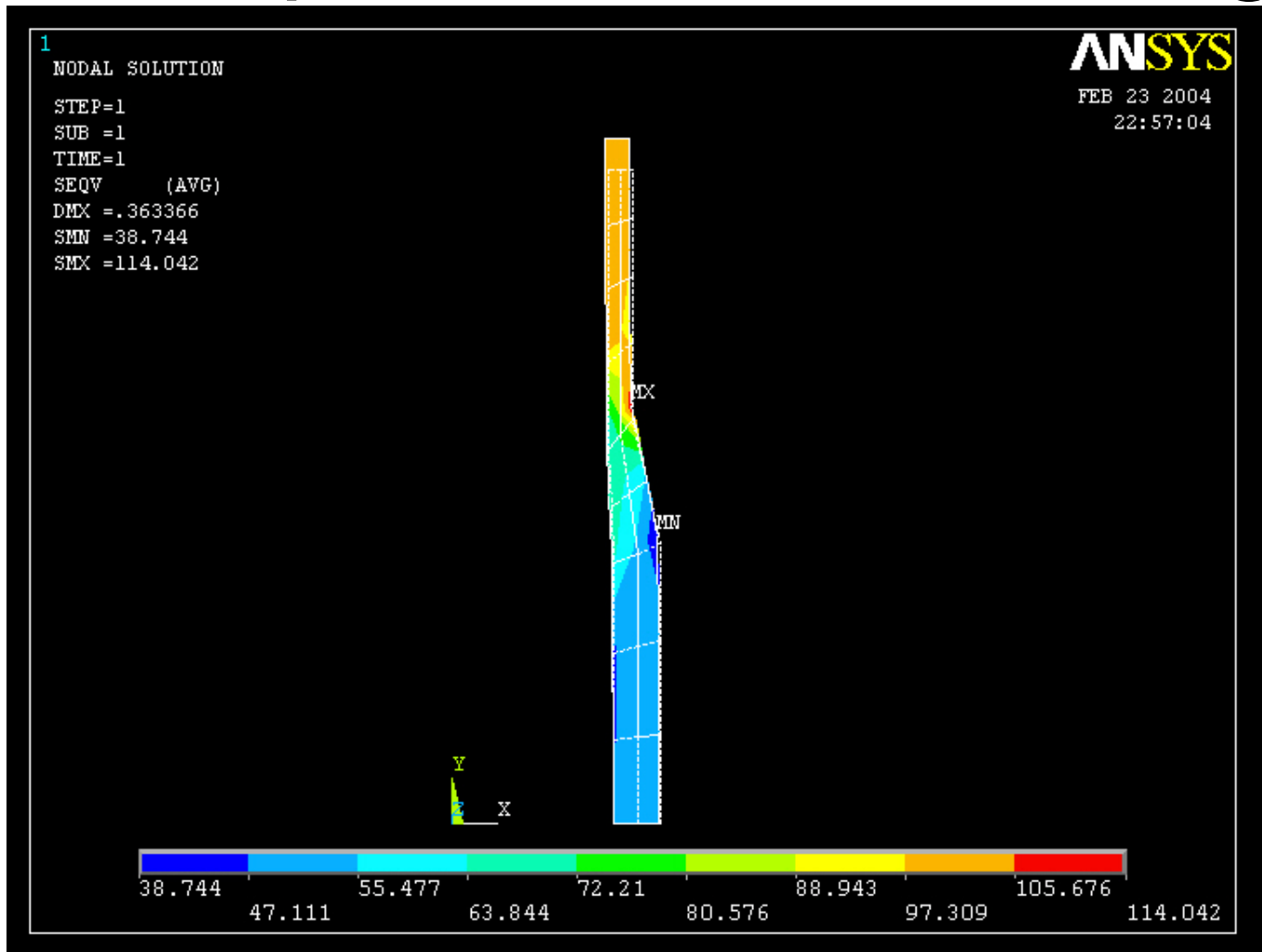
☐ All applicable

[AVPRIN] Eff NU for EQV strain

OK Apply Cancel Help

Select "Def+undeformed" and Press OK

Example - PostProcessing



Read Maximum displacement: DMX

Example0304