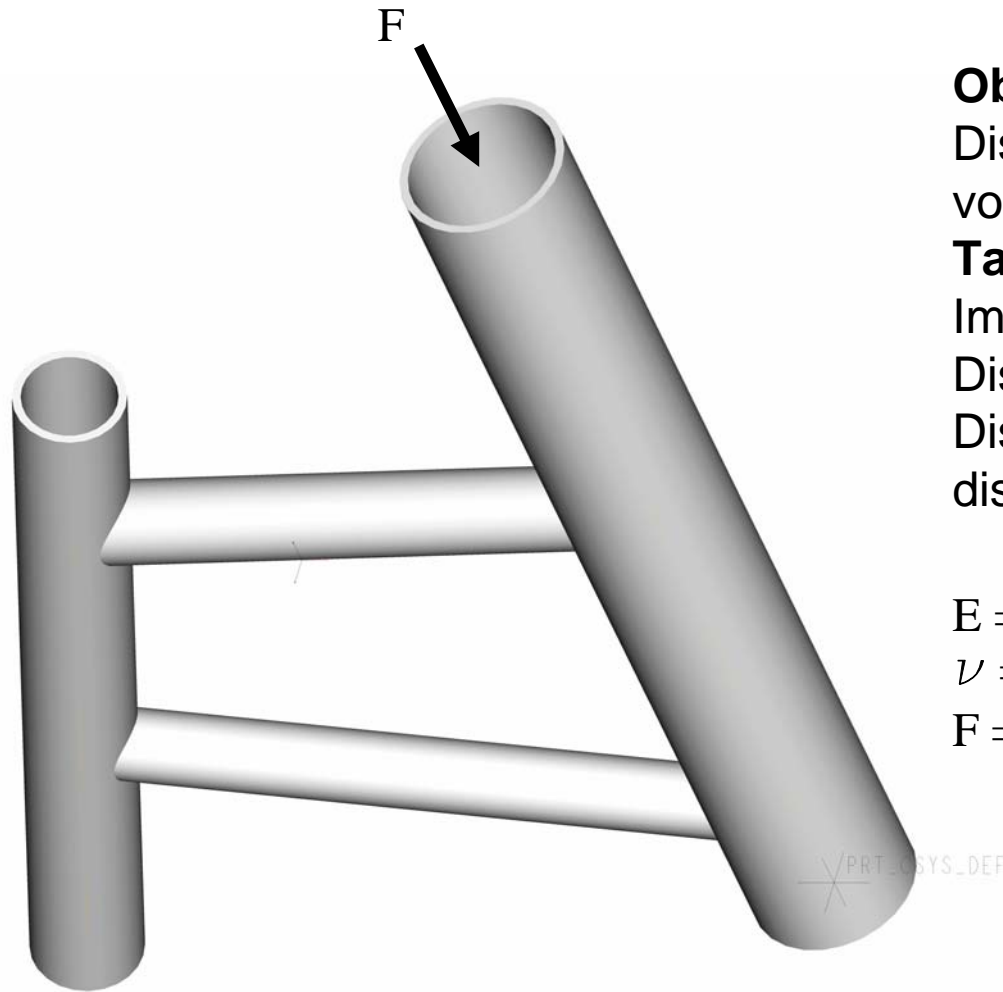


Course in ANSYS

Example0153

Example – Offshore structure



Objective:

Display the deflection figure and von Mises stress distribution

Tasks:

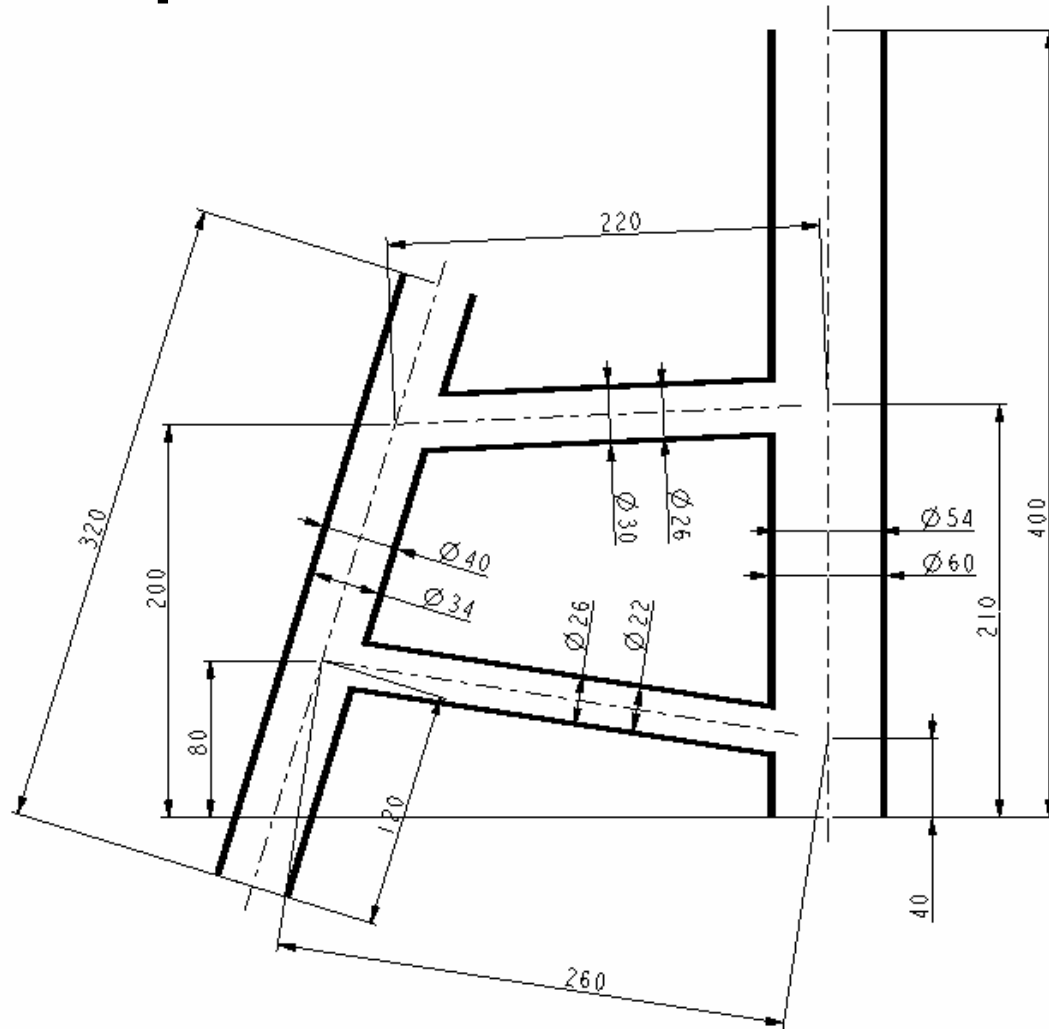
Import geometry from IGES.
Display the deflection figure?
Display the von Mises stress distribution?

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

$$\nu = 0.3$$

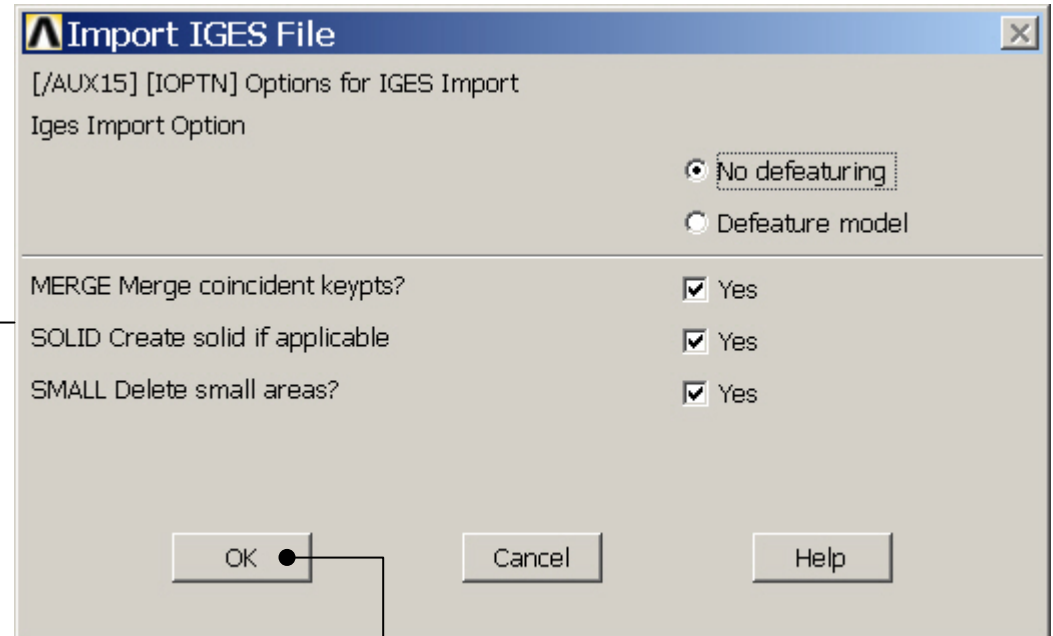
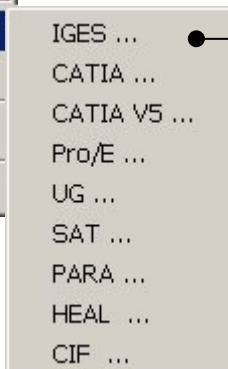
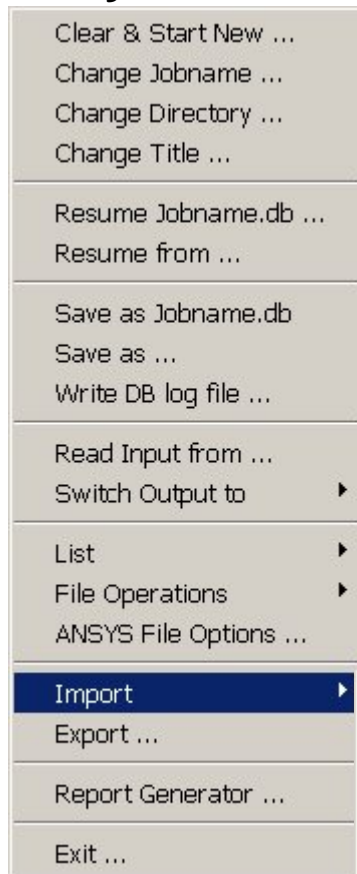
$$F = -10000$$

Example – Offshore structure



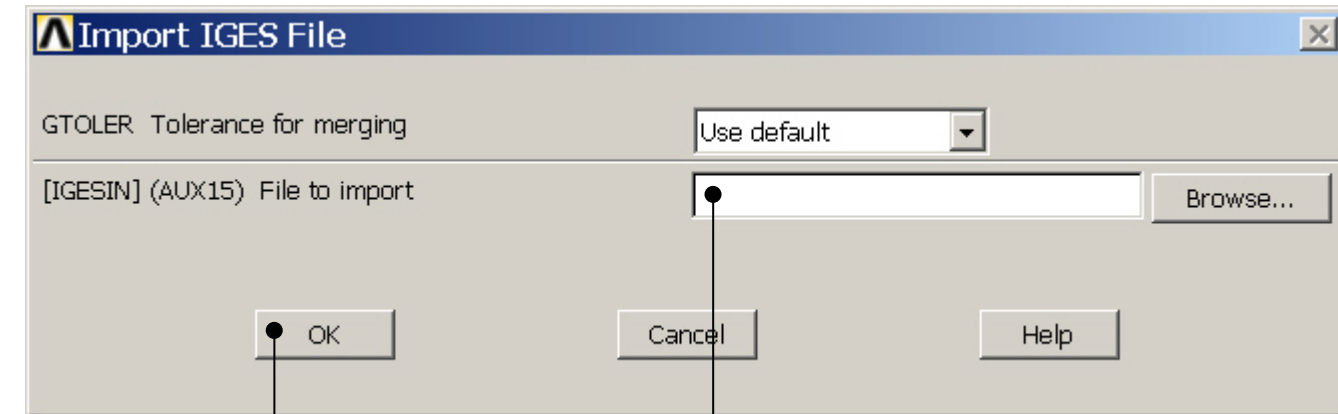
Example – Import IGES

Utility Menu > File > Import > IGES



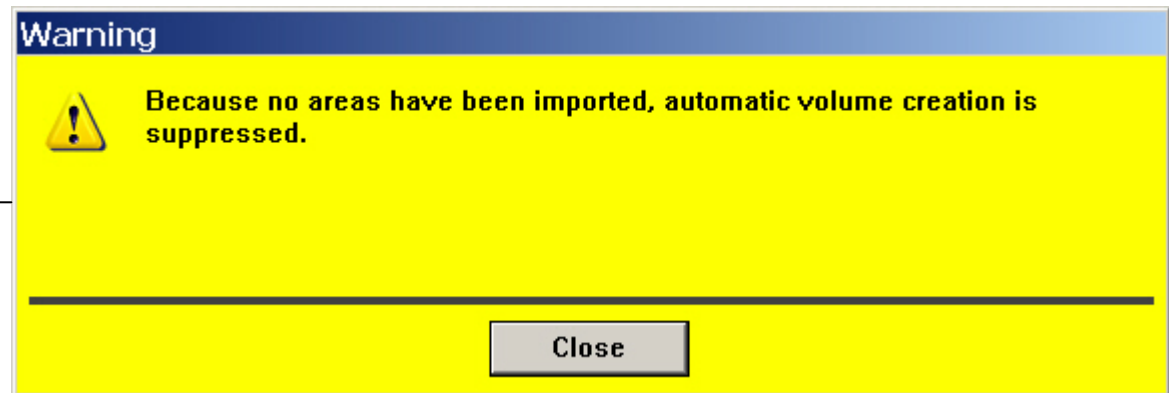
Press OK

Example – Import IGES

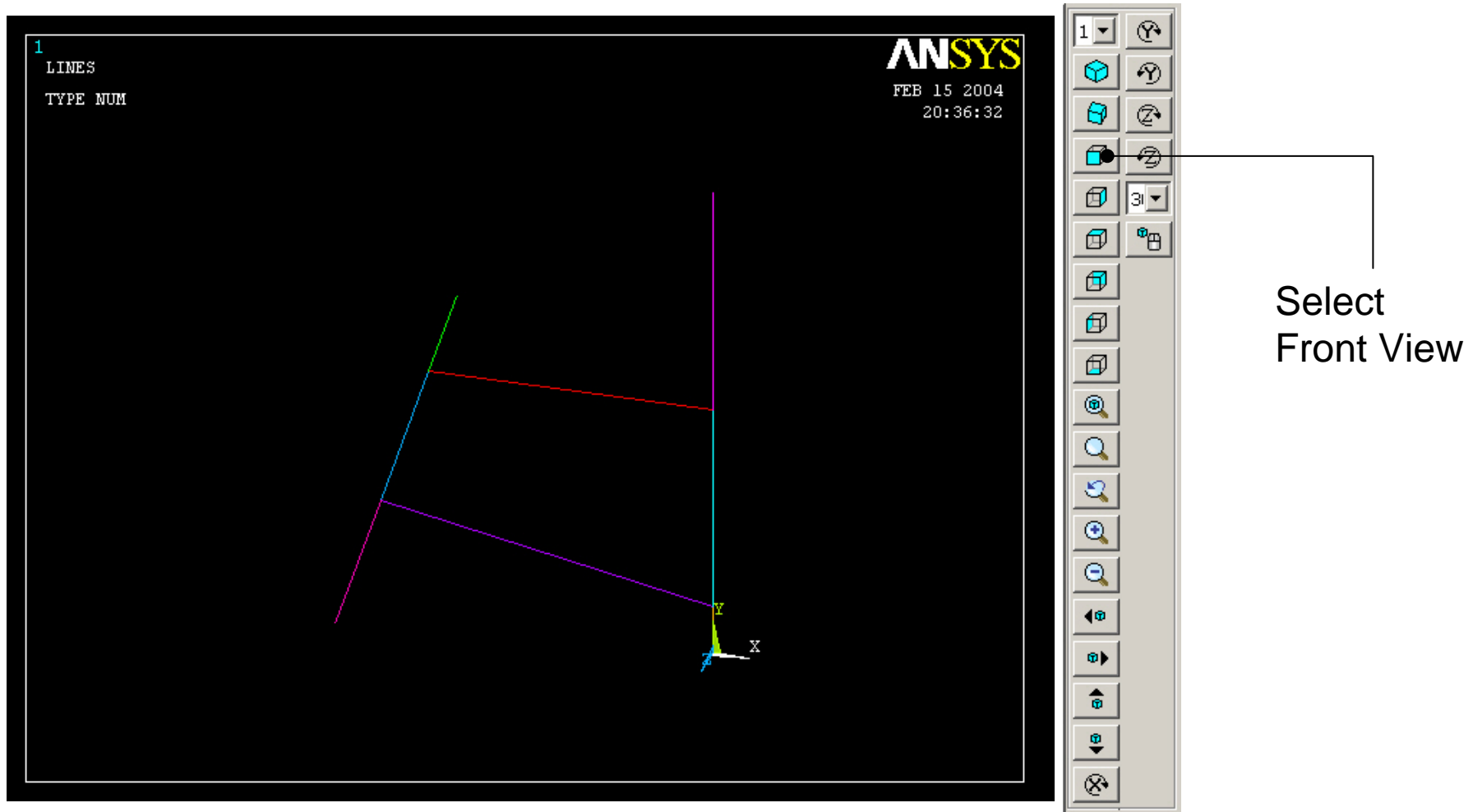


Press OK

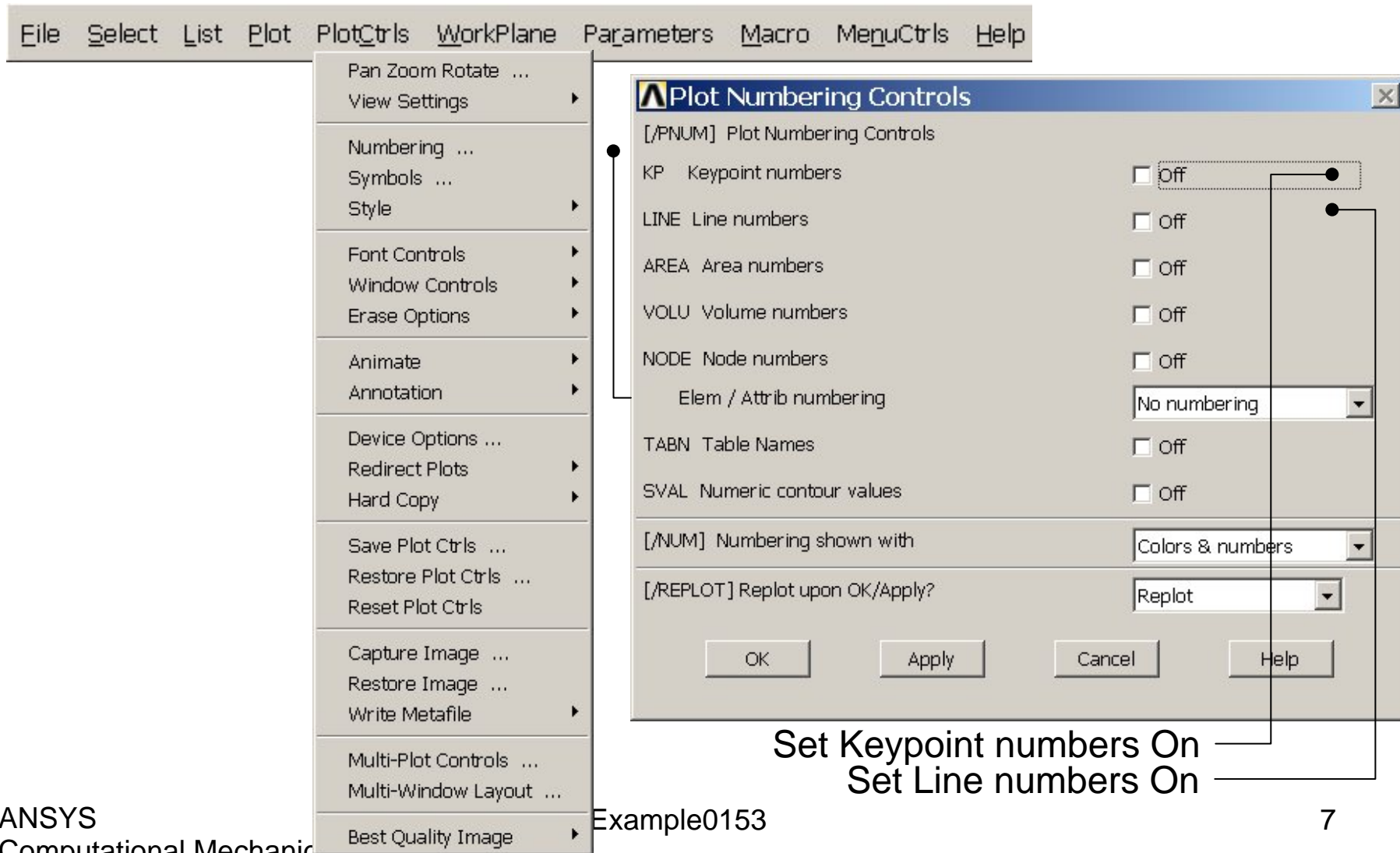
Browse to find offshore-structure-skeleton.igs



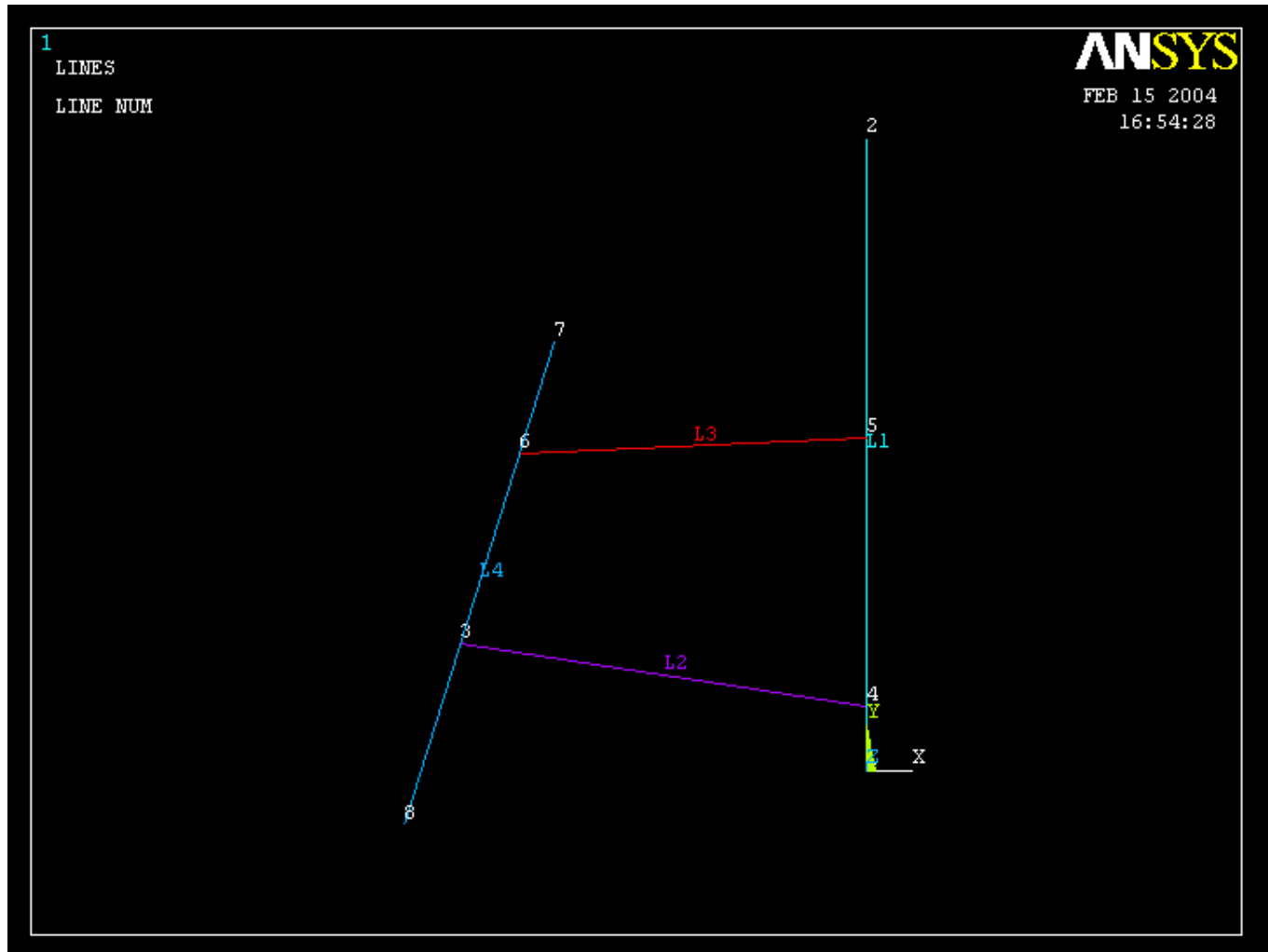
Example - Display



Example - Numbering

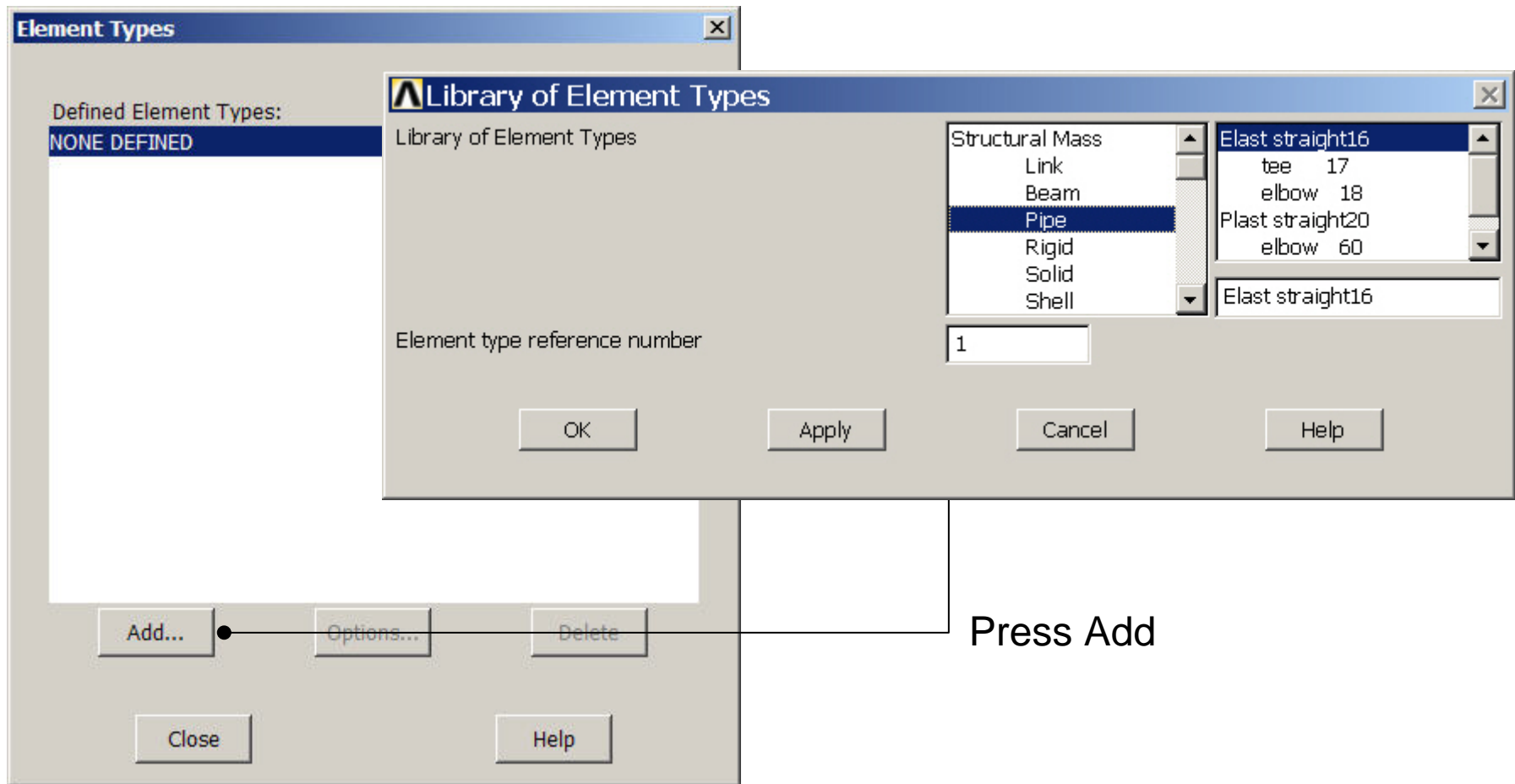


Example - Numbering



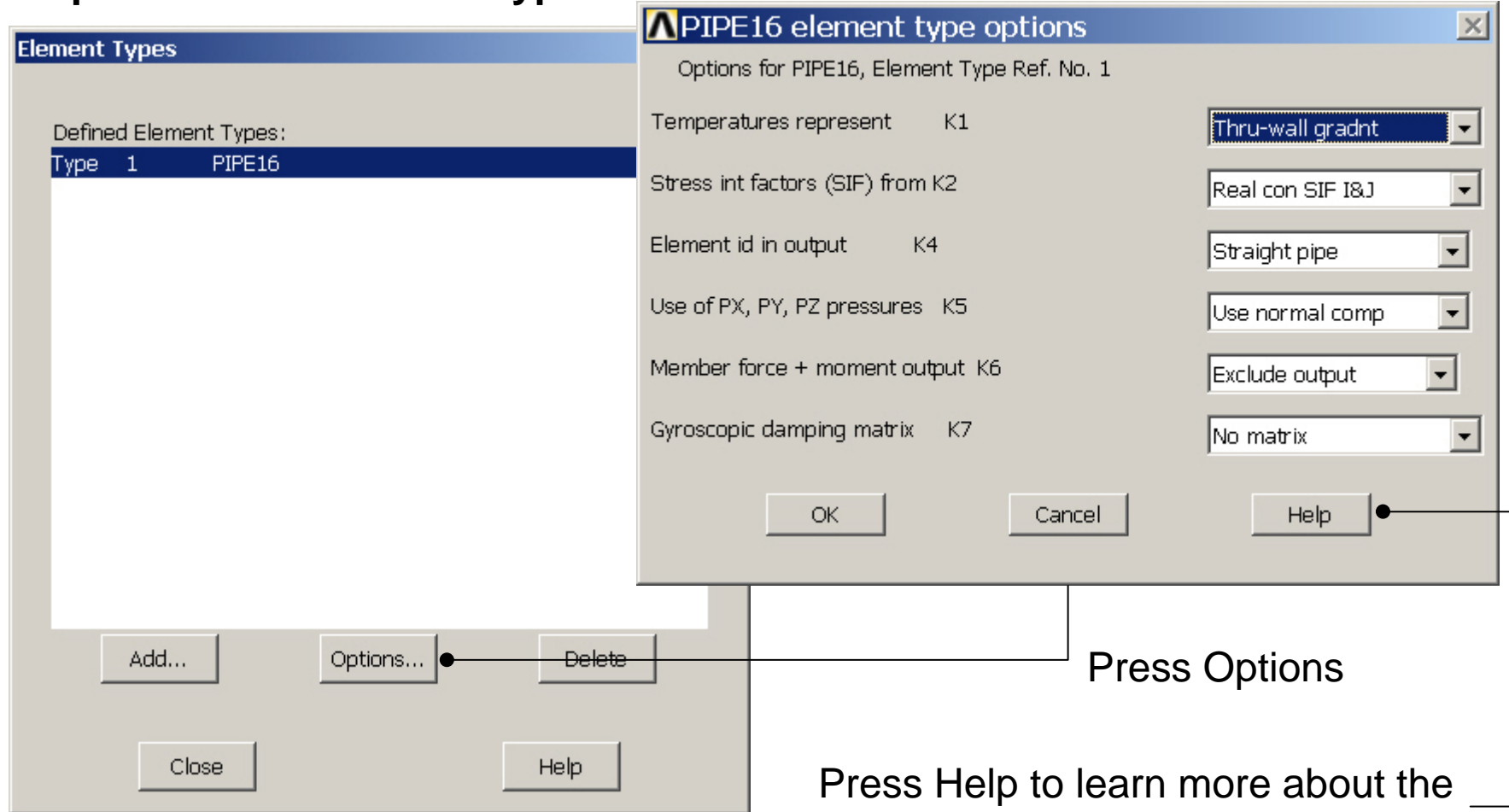
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



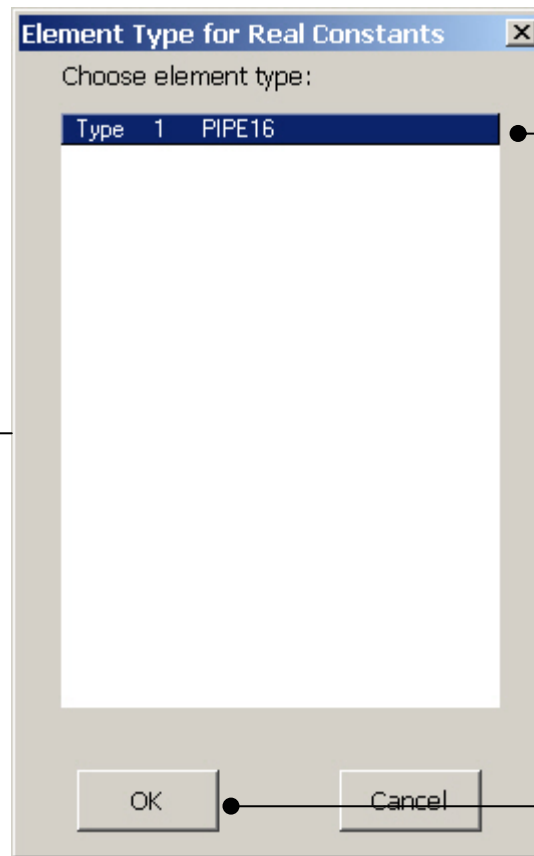
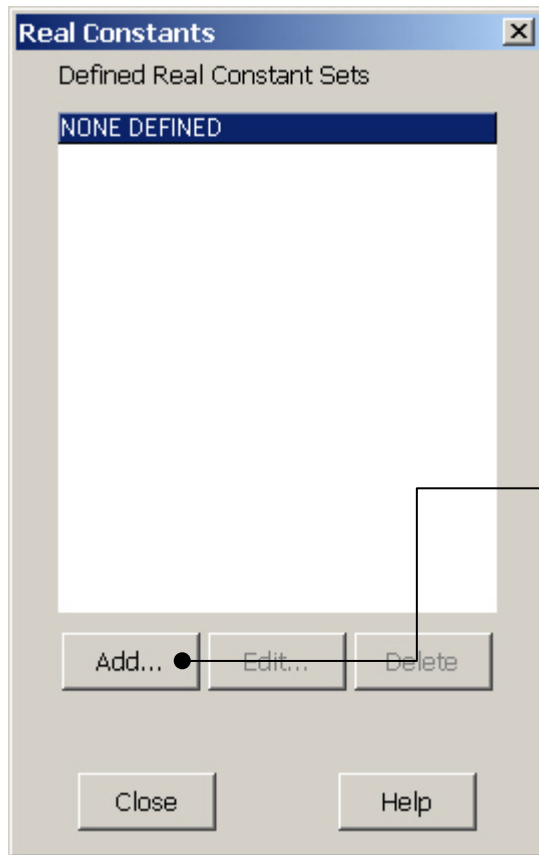
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Real Constants

Preprocessor > Real Constants > Add



Place the cursor
on the relevant
element and
press OK

Example - Real Constants

Preprocessor > Real Constants > Add

The image shows two dialog boxes from the ANSYS Preprocessor. The left dialog, 'Real Constant Set Number 1, for PIPE16', contains fields for various material properties, all set to 0. The right dialog, 'Real Constants', shows a list of four defined sets. Annotations with arrows point to specific elements: 'Enter OD=60 and TKWALL=3' points to the first set; 'Enter OD=40 and TKWALL=3' points to the second; 'Enter OD=26 and TKWALL=2' points to the third; 'Enter OD=30 and TKWALL=2' points to the fourth; 'Add 4 Sets' points to the 'Add...' button; 'Press Close to finish' points to the 'Close' button; and 'Press OK' points to the 'OK' button in the left dialog.

Real Constant Set Number 1, for PIPE16

Element Type Reference No. 1
Real Constant Set No. 1

Outside diameter OD 0
Wall thickness TKWALL 0
Stress intensity fact at I SIFI 0
Stress intensity fact at J SIFJ 0
Flexibility factor FLEX 0
Internal fluid density DENSFL 0
Ext insulation density DENSIN 0
Insulation thickness TKIN 0
Corrosion thk allowance TKCORR 0
Insulation surface area AREAIN 0
Pipe wall mass MWALL 0
Axial pipe stiffness STIFF 0
Rotordynamic spin SPIN 0

OK Apply Cancel Help

Real Constants

Defined Real Constant Sets

Set	1
Set	2
Set	3
Set	4

Add... Edit... Delete

Close Help

Enter OD=60 and TKWALL=3
Enter OD=40 and TKWALL=3
Enter OD=26 and TKWALL=2
Enter OD=30 and TKWALL=2

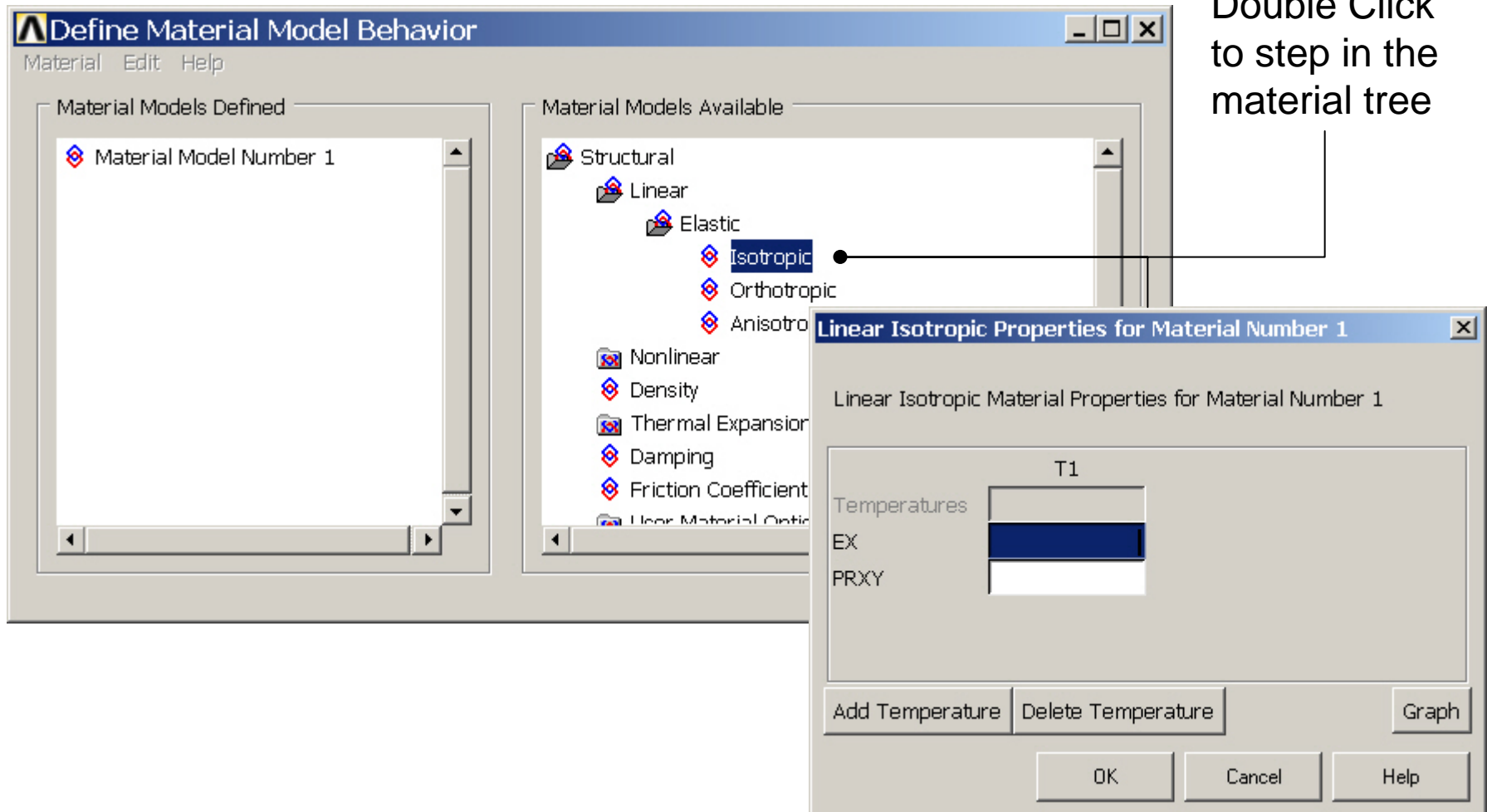
Add 4 Sets

Press Close to finish

Press OK

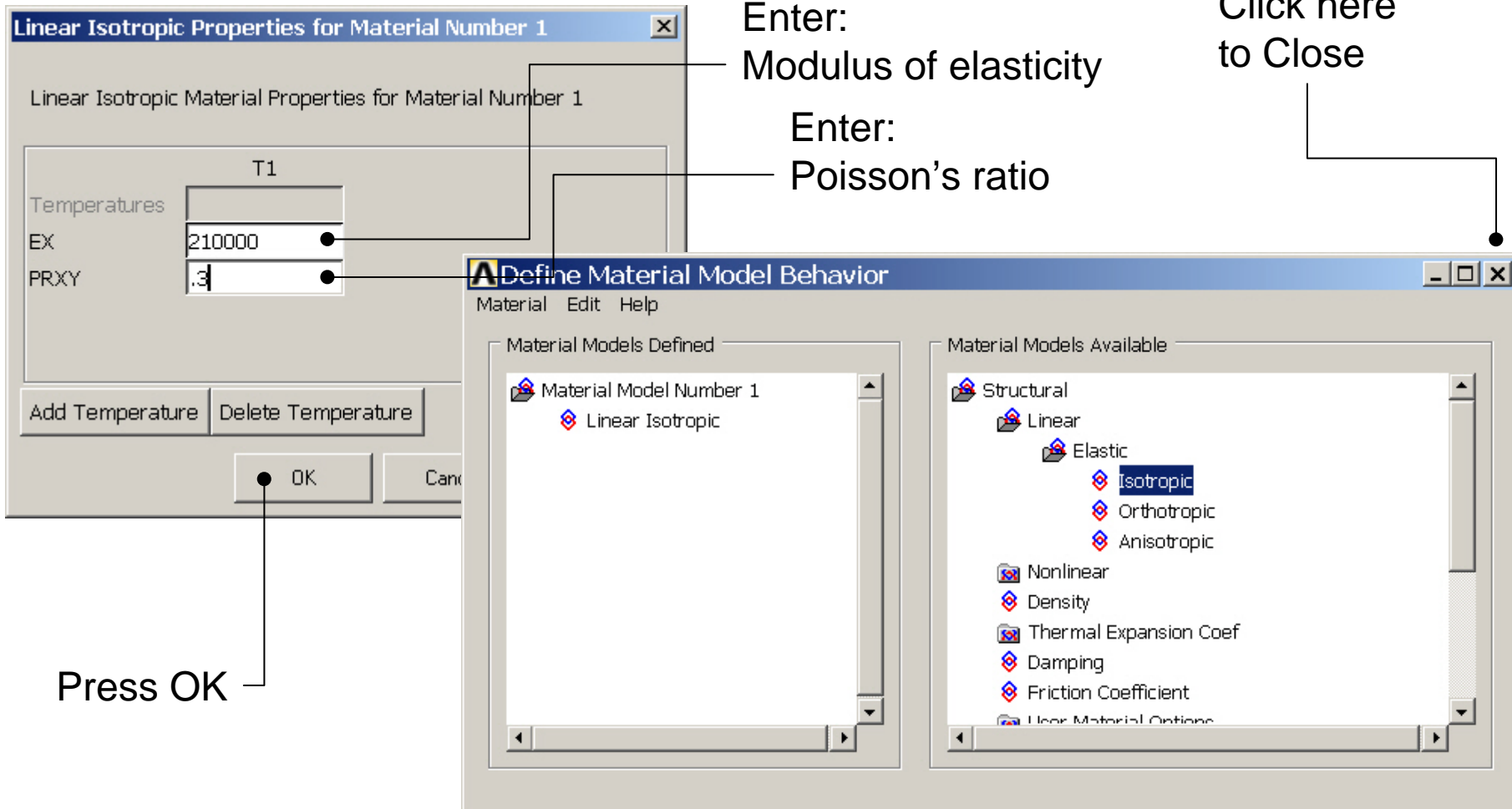
Example - Material Properties

Preprocessor > Material Props > Material Models



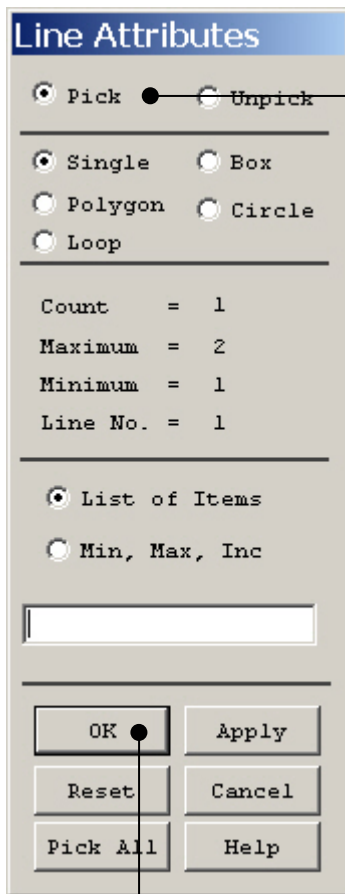
Example - Material Properties

Preprocessor > Material Props > Material Models

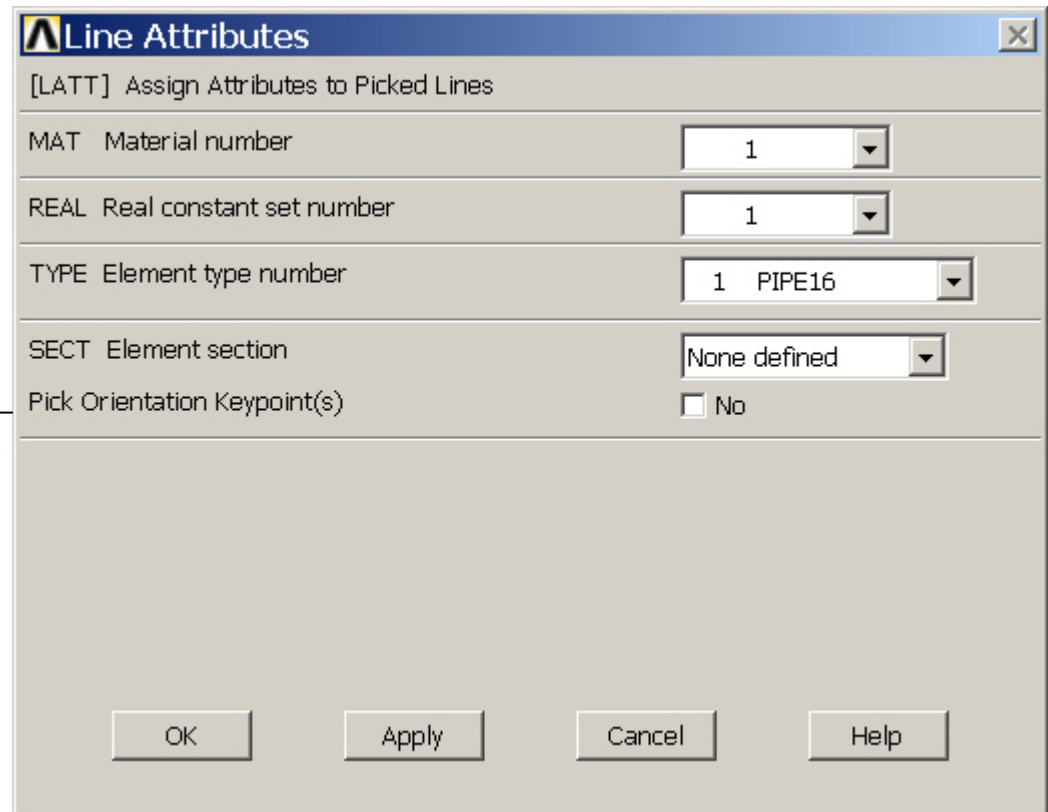


Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines



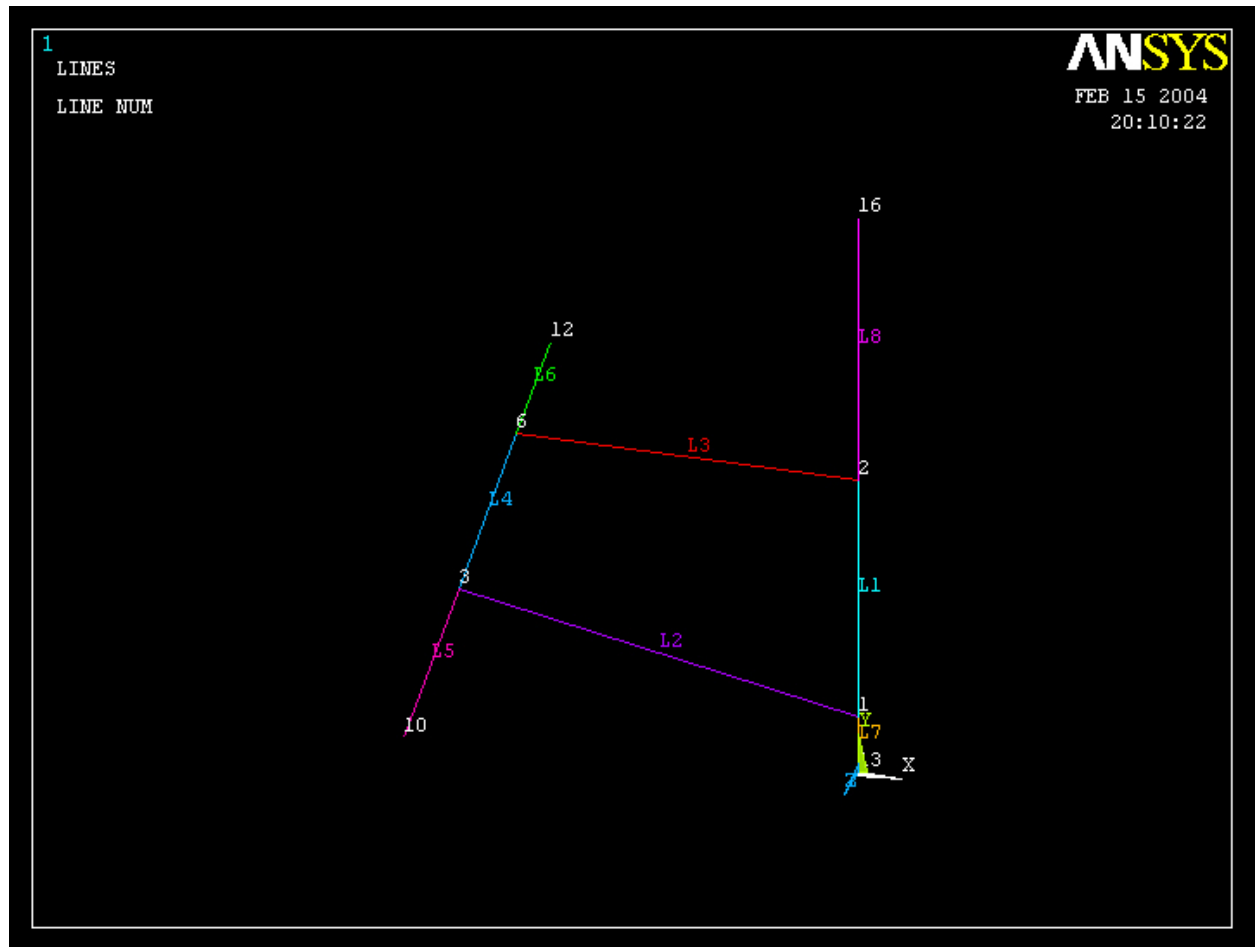
Select Line L7, L1, L8



ANSYS Press OK
Computational Mechanics, AAU, Esbjerg

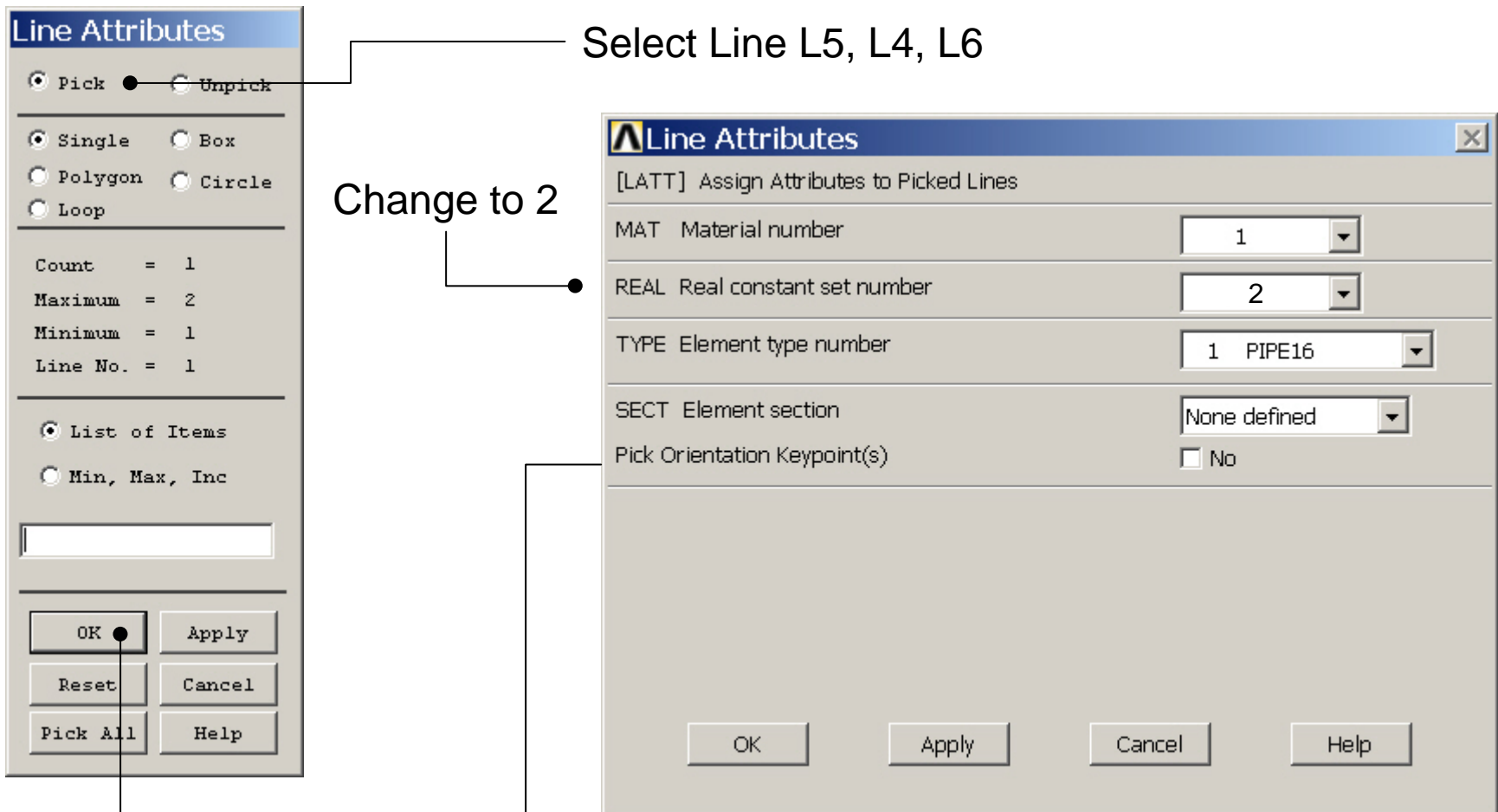
Example0153

Example – Mesh Attributes

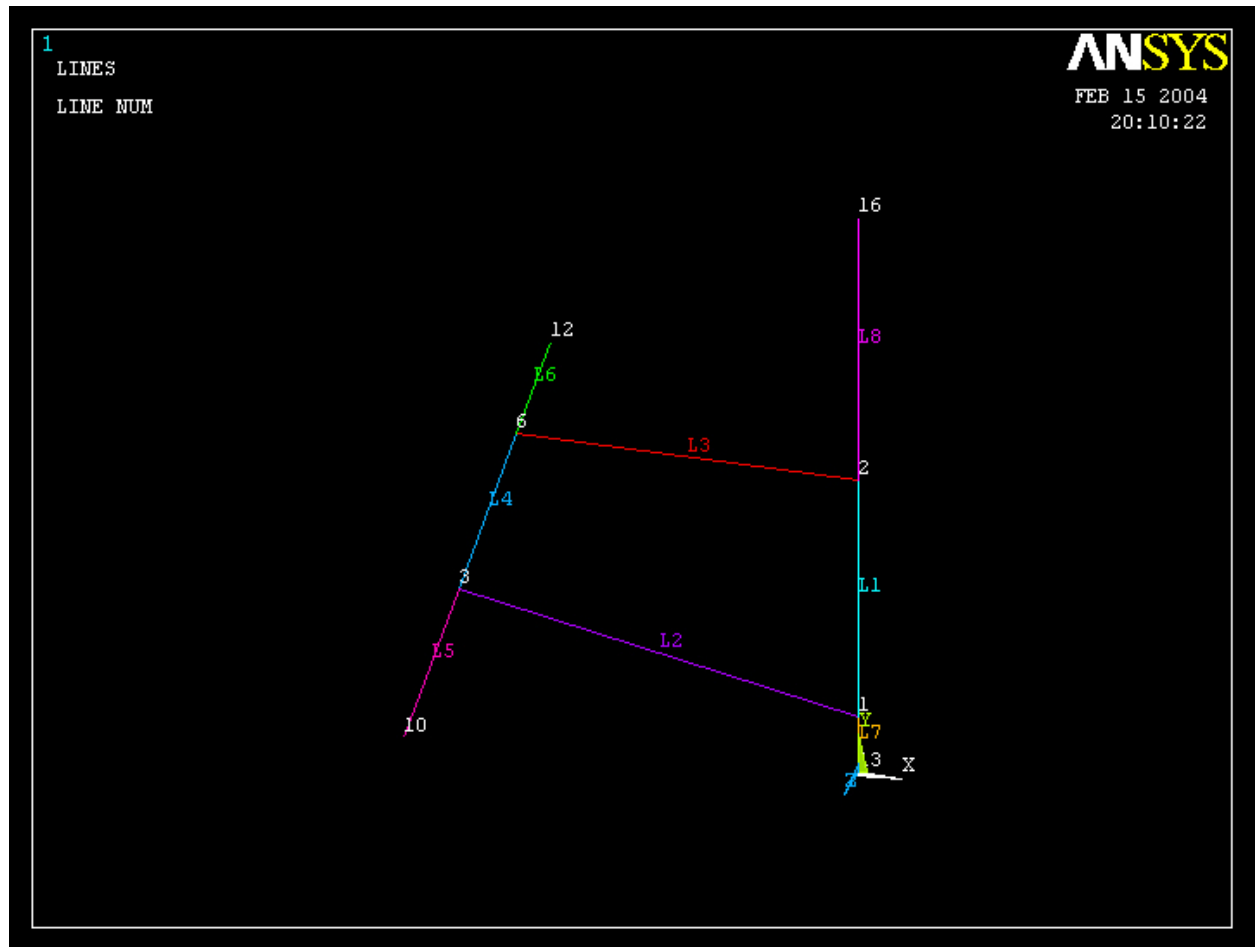


Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines

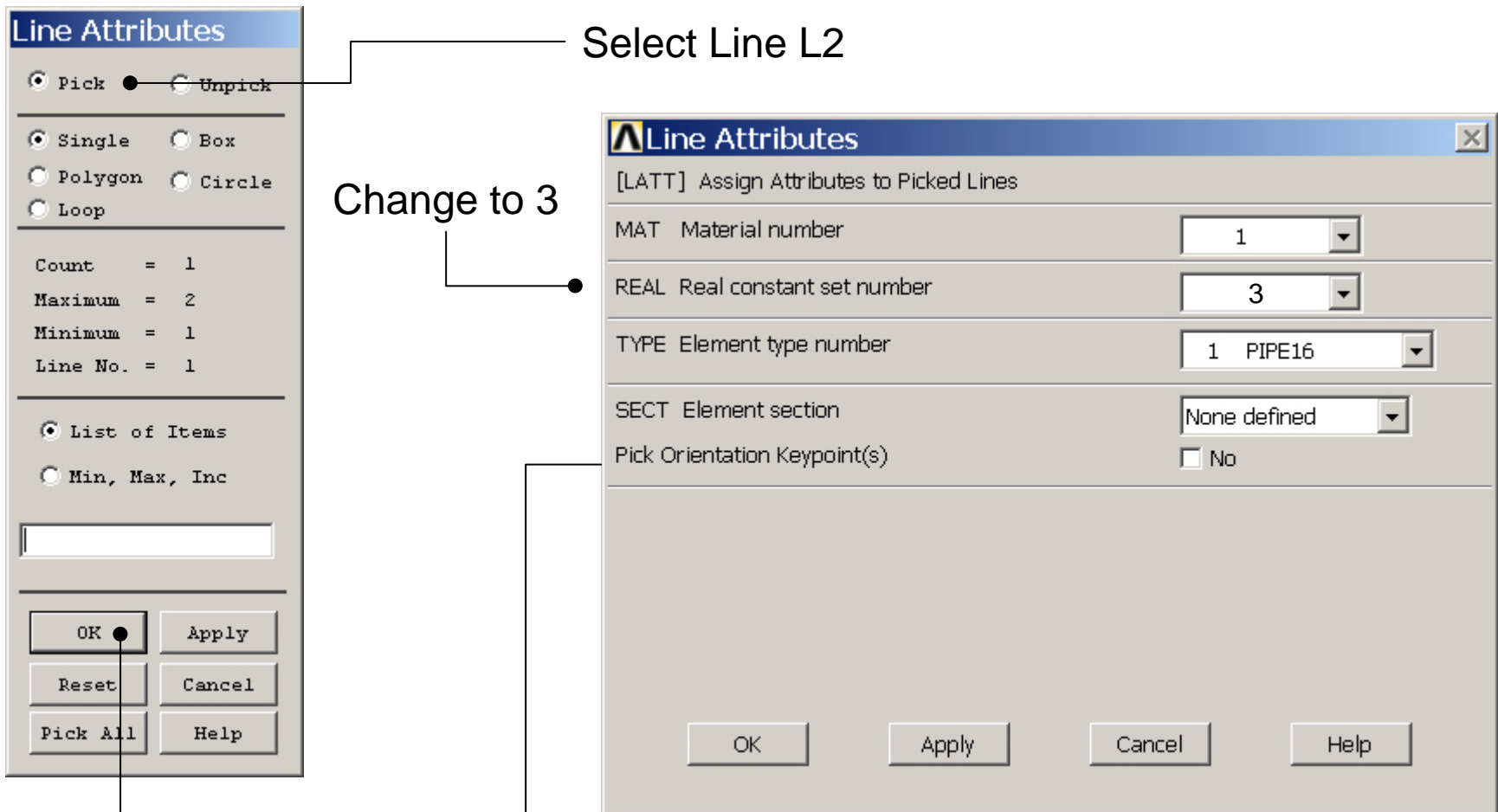


Example – Mesh Attributes

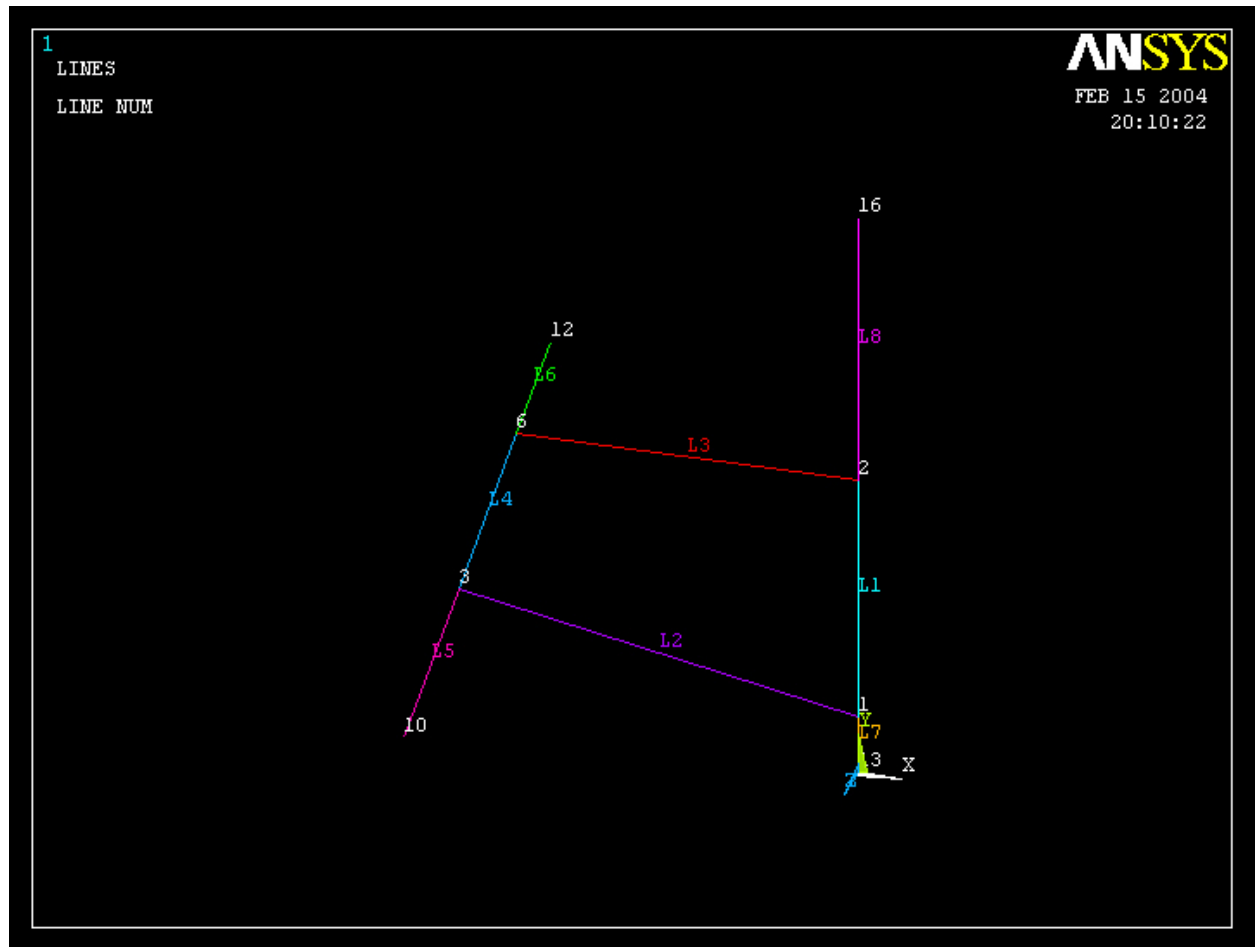


Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines

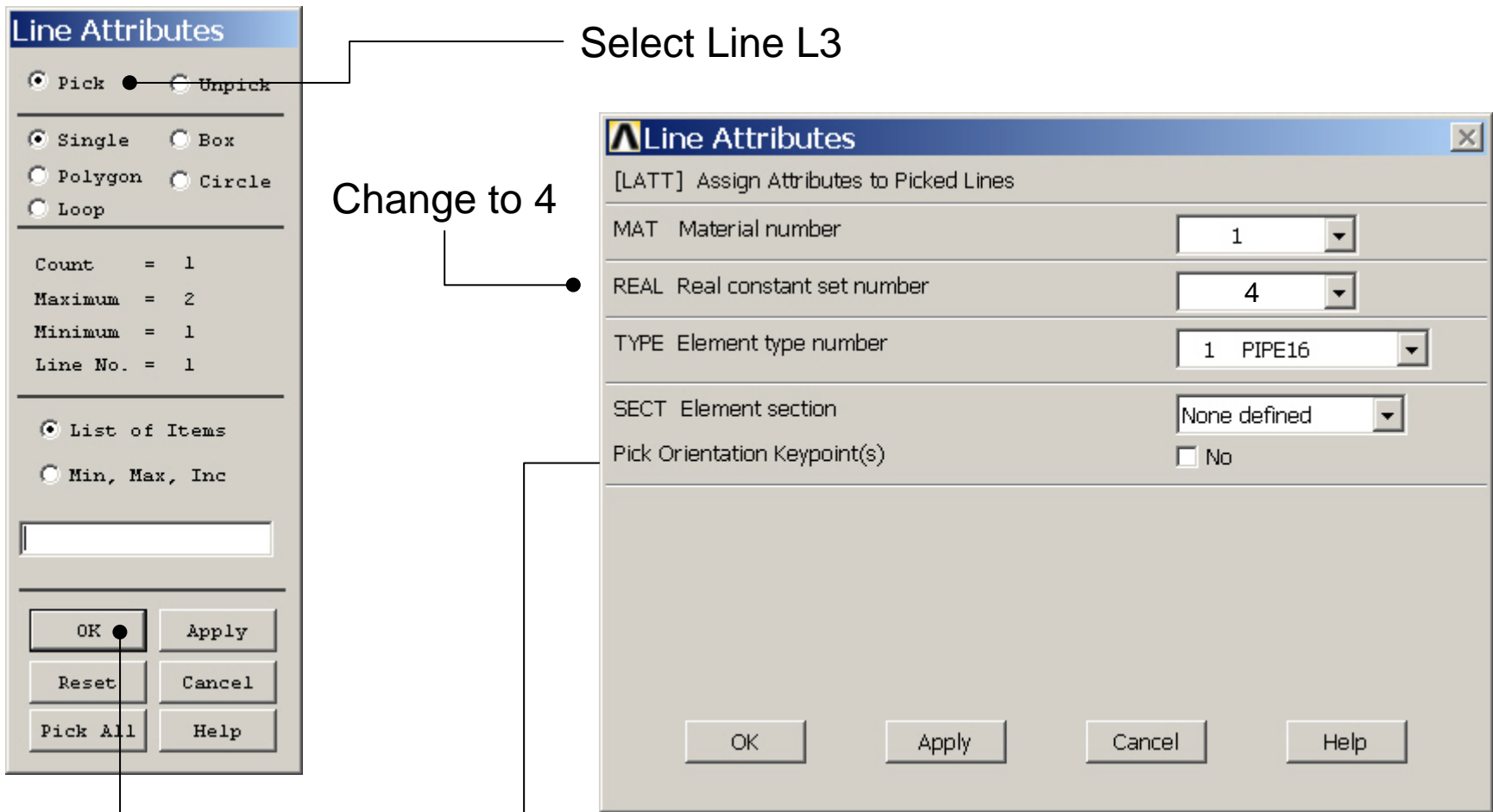


Example – Mesh Attributes

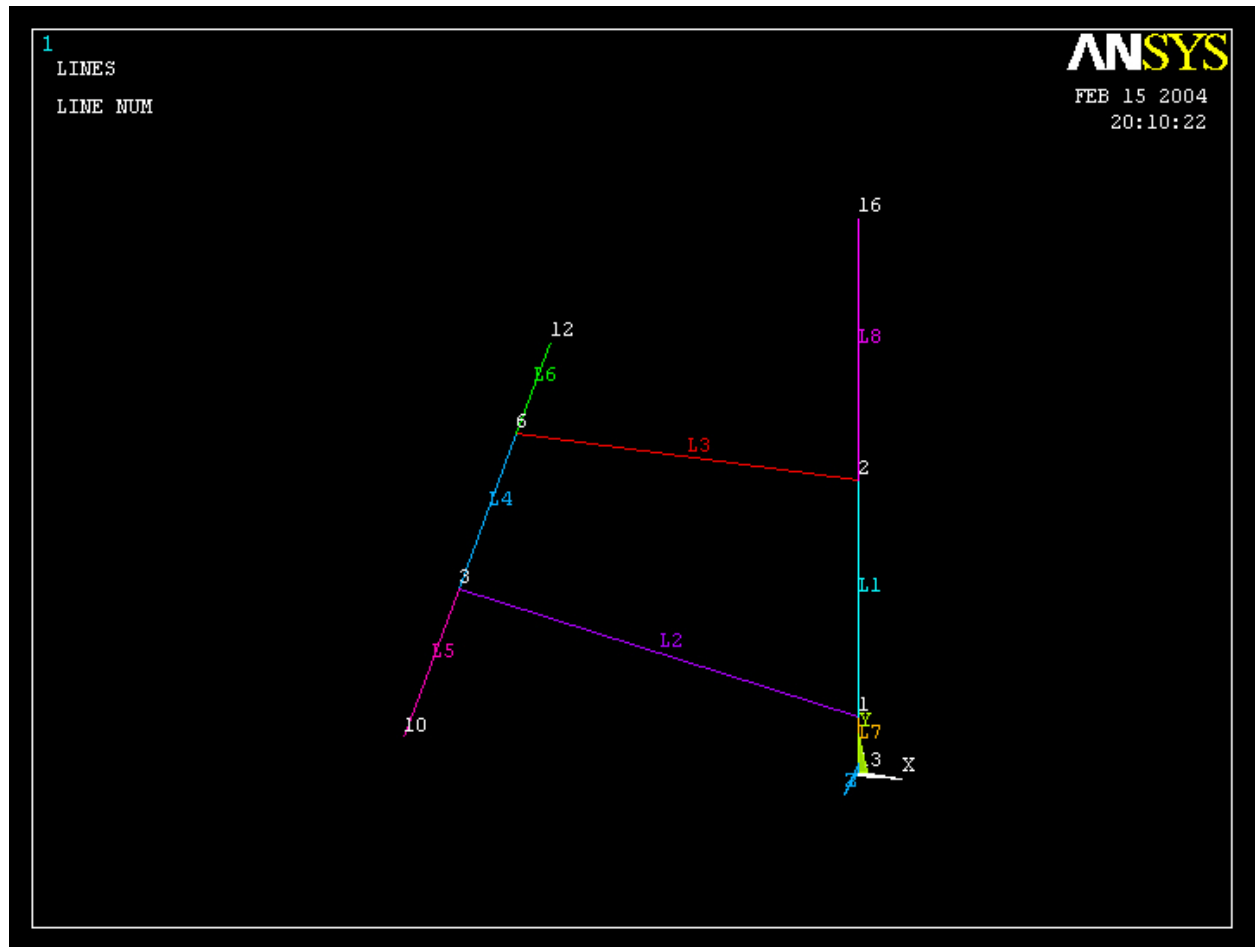


Example – Mesh Attributes

Preprocessor > Meshing > Mesh Attributes > Line Attributes > Picked Lines



Example – Mesh Attributes



Example - Meshing

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

Select/Pick
Lines to
specify
mesh size
for

Element Size on P...

☒ Pick ☐ Unpick

☒ Single ☐ Box

☐ Polygon ☐ Circle

☐ Loop

Count = 0

Maximum = 1

Minimum = 1

Line No. =

☒ List of Items

☐ Min, Max, Inc

OK Apply

Reset Cancel

Pick All Help

Element Sizes on Picked Lines

[LESIZE] Element sizes on picked lines

SIZE Element edge length

NDIV No. of element divisions

(NDIV is used only if SIZE is blank or zero)

KYNDIV SIZE,NDIV can be changed ☒ Yes

SPACE Spacing ratio

ANGSIZ Division arc (degrees)

(use ANGSIZ only if number of divisions (NDIV) and element edge length (SIZE) are blank or zero)

Clear attached areas and volumes ☐ No

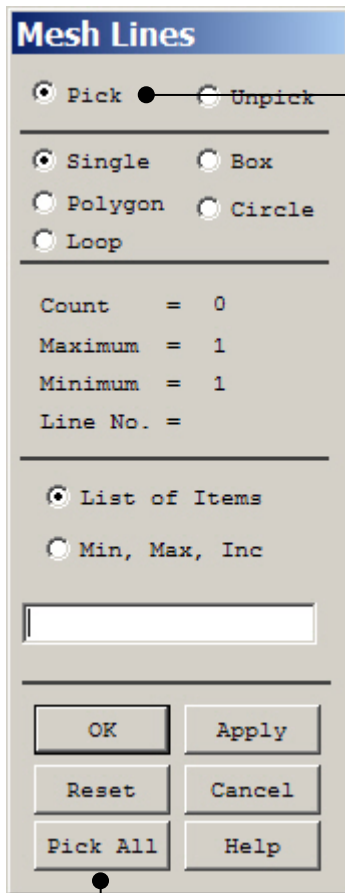
OK Apply Cancel

Press OK when finish with selection

Enter
2 for L7
3 for L6
4 for L1,L4,L5
5 for L8

Example - Meshing

Preprocessor > Meshing > Mesh > Lines

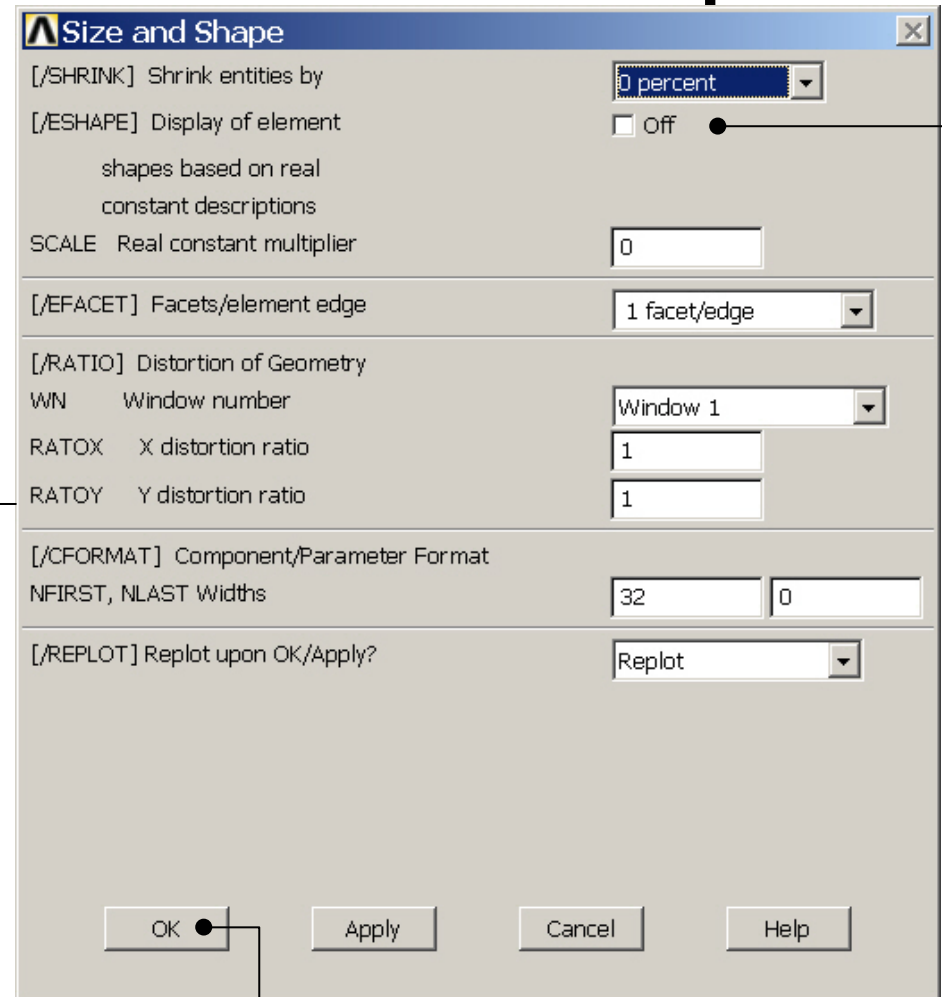
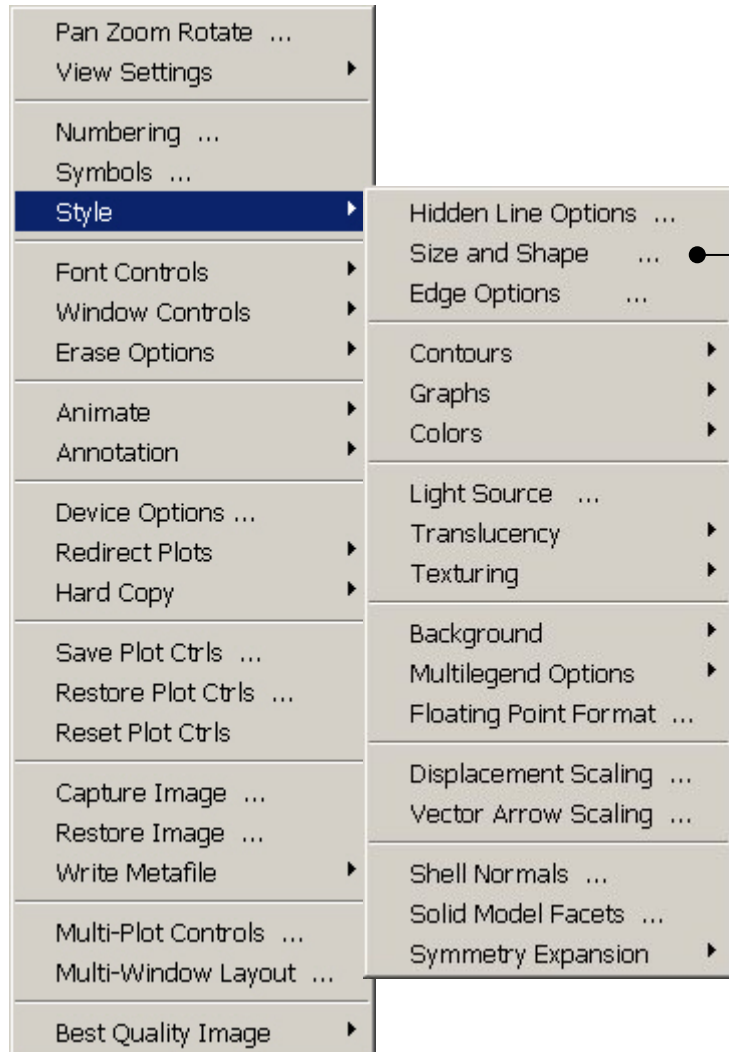


Select individual lines to be meshed by Picking

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

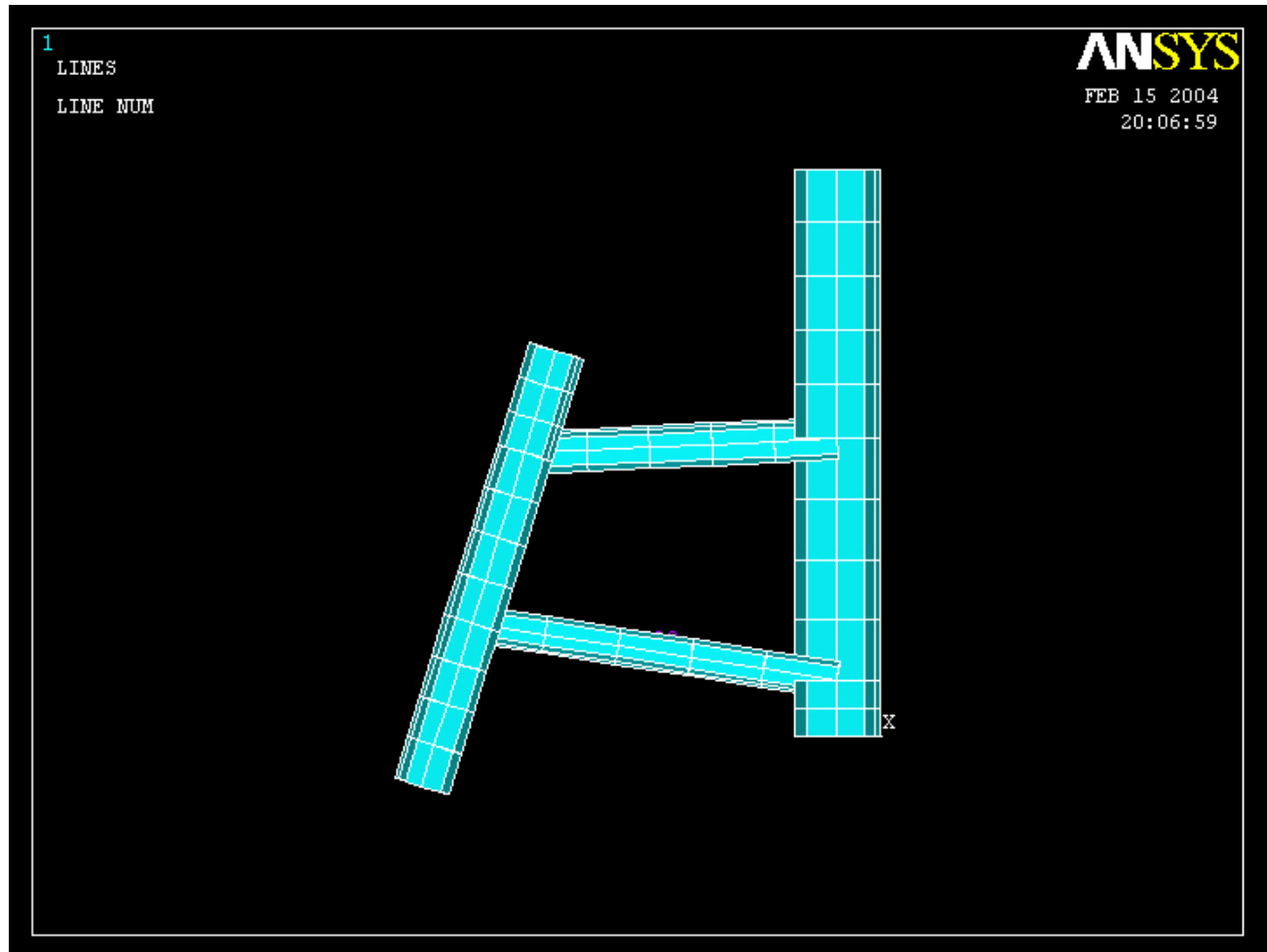
Select all lines defined to be meshed

Example – Size and Shape



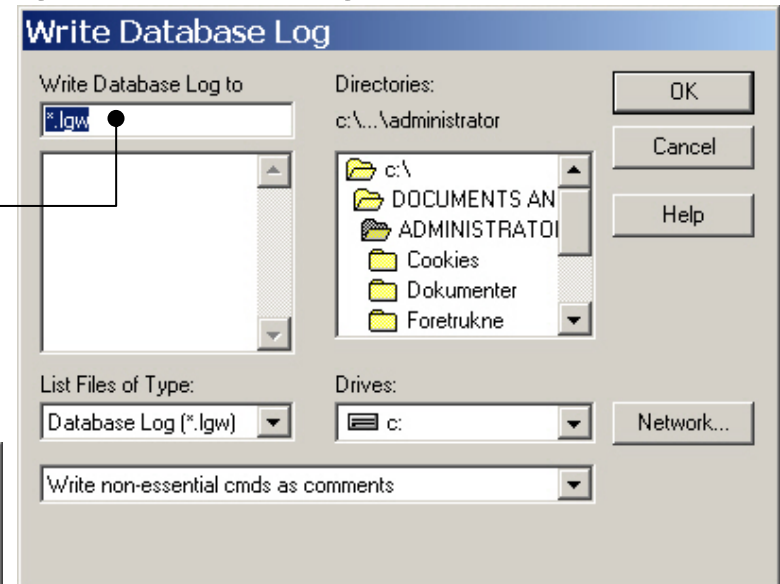
Press OK — Switch to On —

Example – Display of element

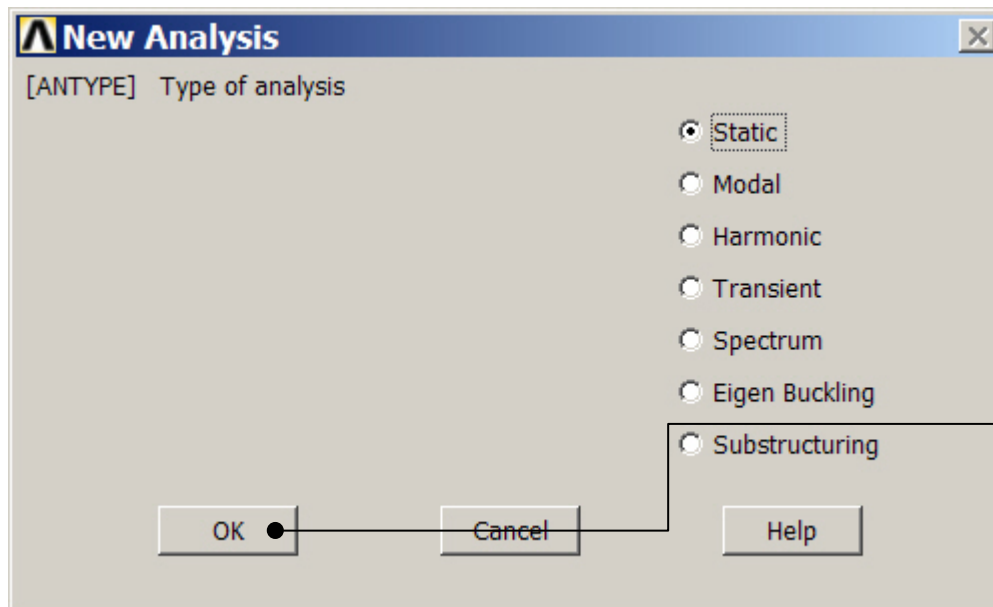


Example – Analysis Type

File > Write DB log file
Enter “example0153.lgw”



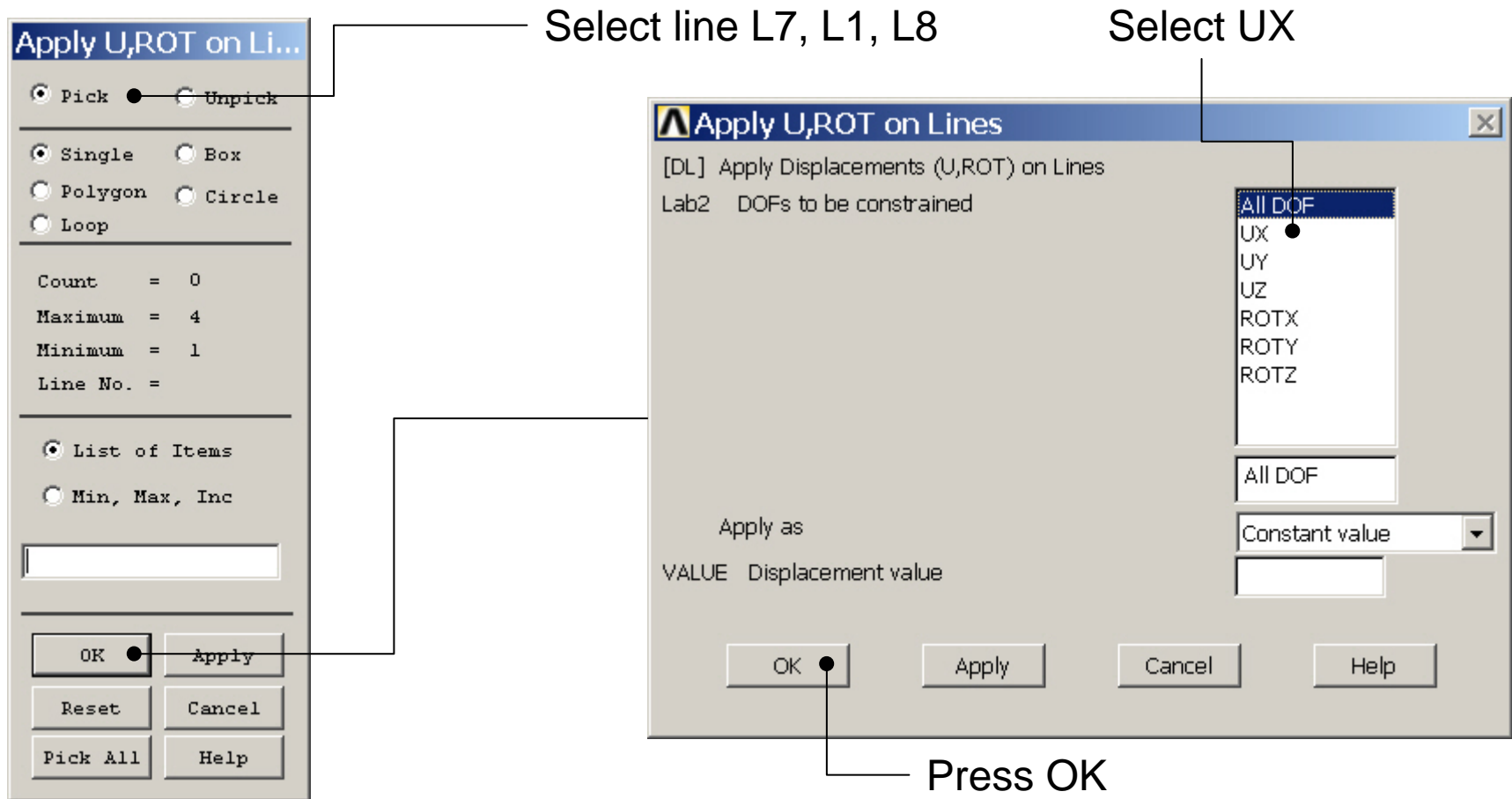
Solution > Analysis Type > New Analysis



Press OK

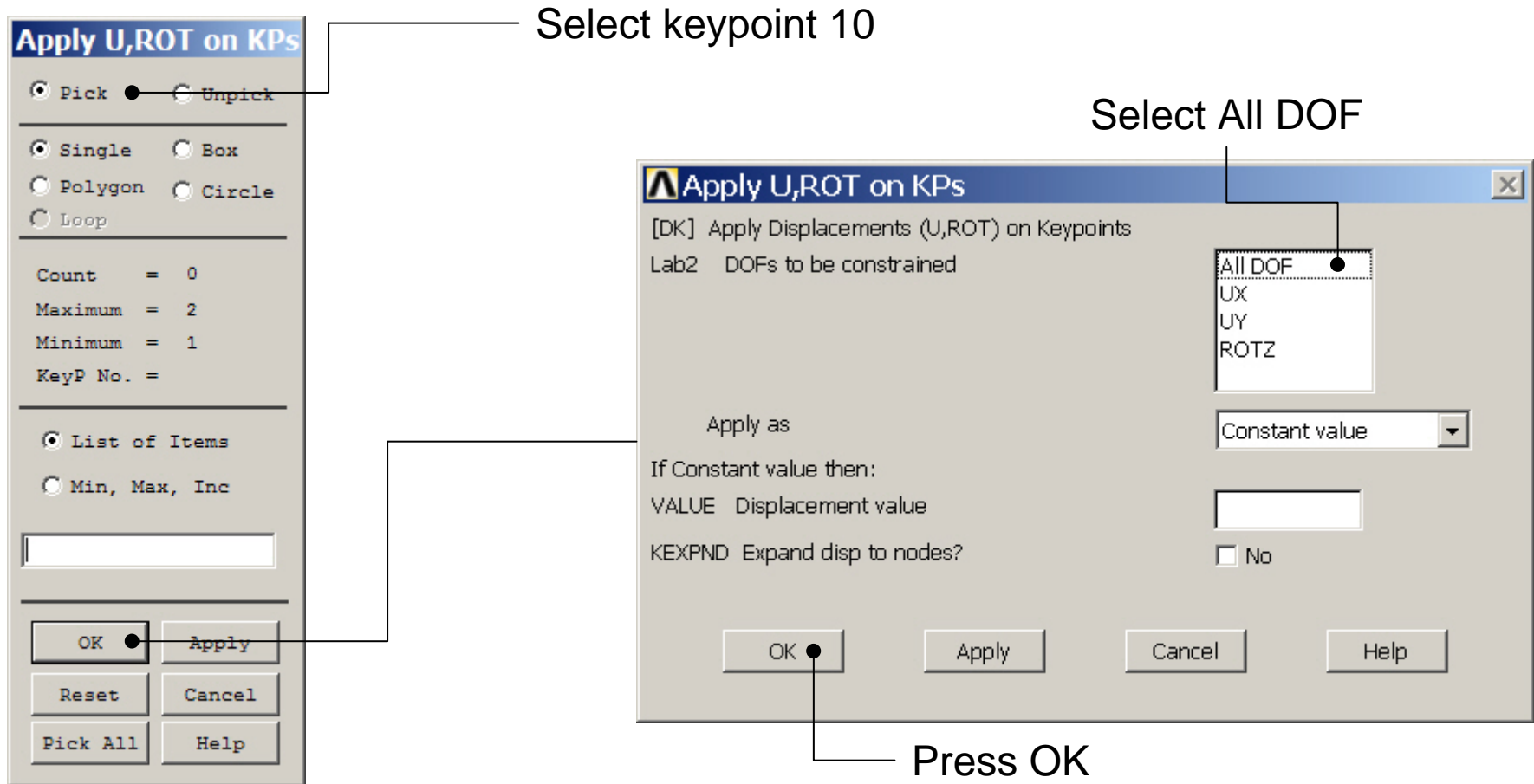
Example – Define Loads

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



Example – Define Loads

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



Example – Define Loads

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

Apply F/M on KPs

☒ Pick ☐ Unpick

☒ Single ☐ Box

☐ Polygon ☐ Circle

☐ Loop

Count = 0

Maximum = 2

Minimum = 1

KeyP No. =

☒ List of Items

☐ Min, Max, Inc

OK Apply

Reset Cancel

Pick All Help

Select keypoint 16

Apply F/M on KPs

[FK] Apply Force/Moment on Keypoints

Lab Direction of force/mom

Apply as

If Constant value then:

VALUE Force/moment value

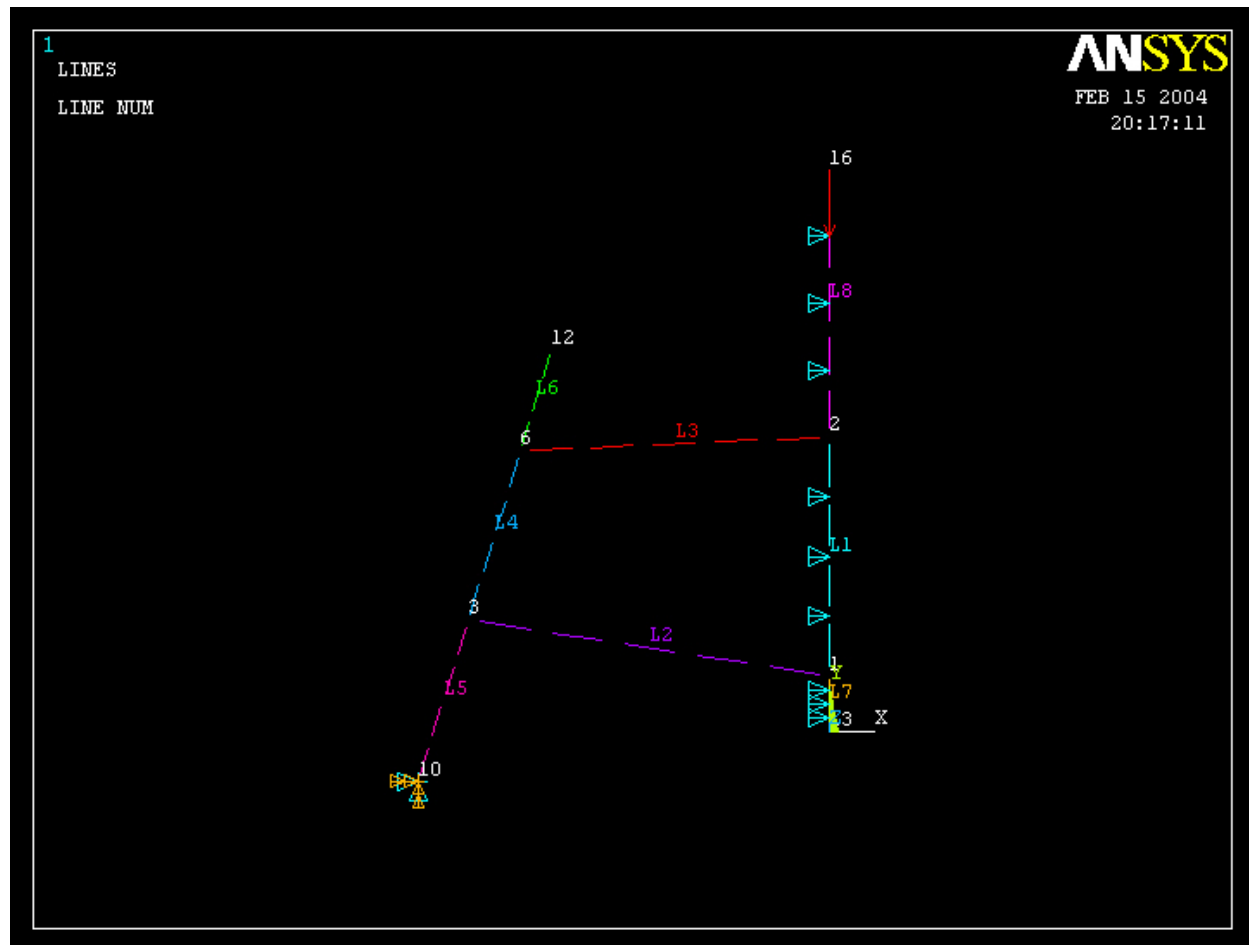
Change to FY

OK Apply Cancel Help

Press OK to finish

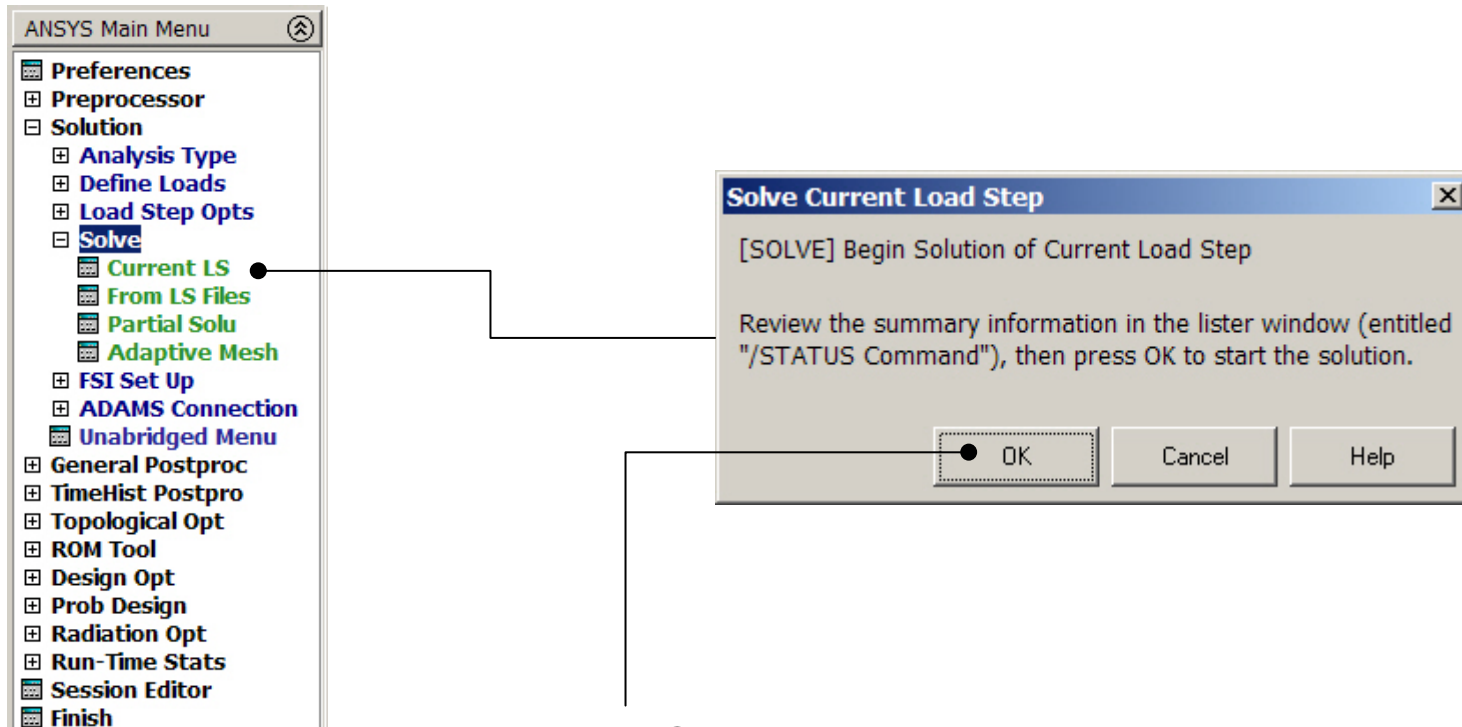
Enter -10000

Example – Boundary Conditions



Example - Solve

Solution > Solve > Current LS

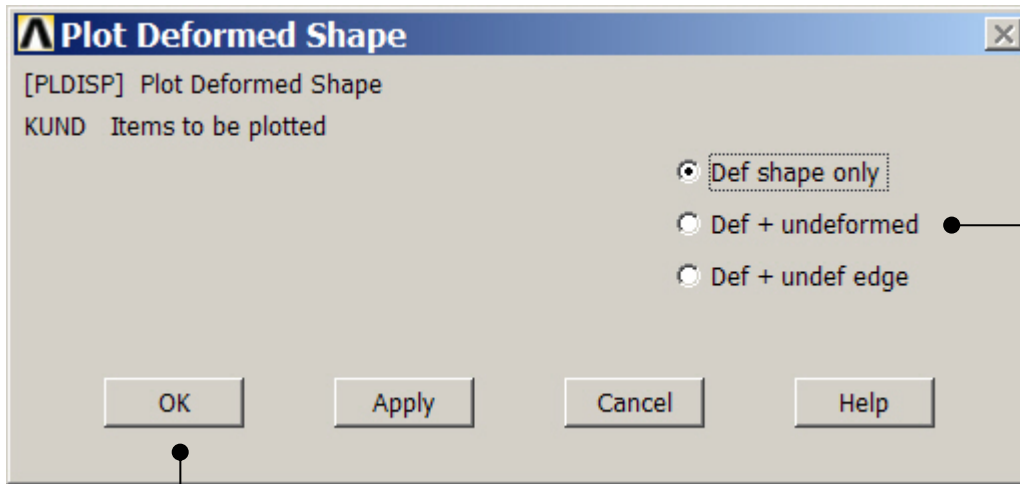


Press OK

Example0153

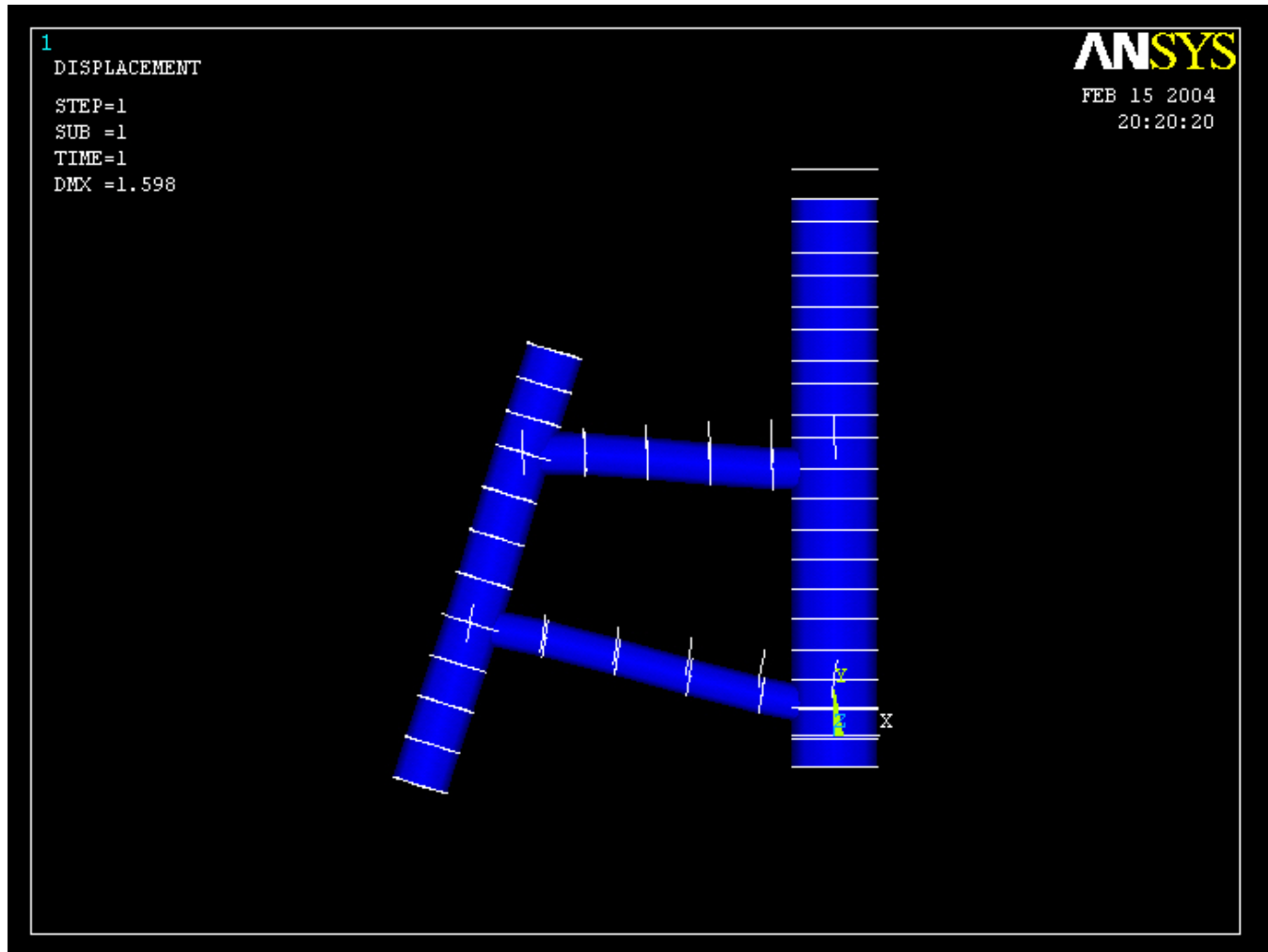
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape

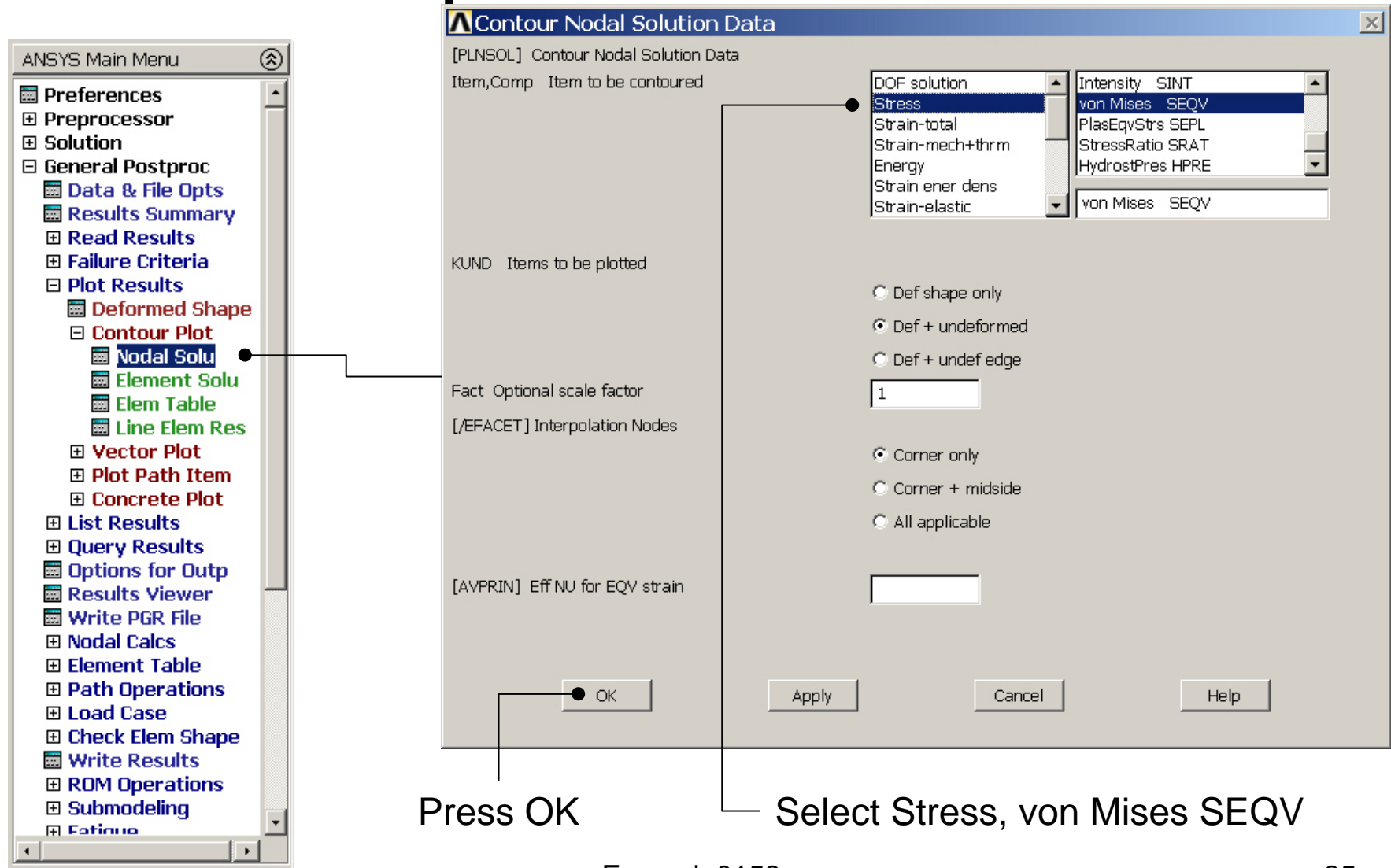


Select "Def+undeformed"
and Press OK

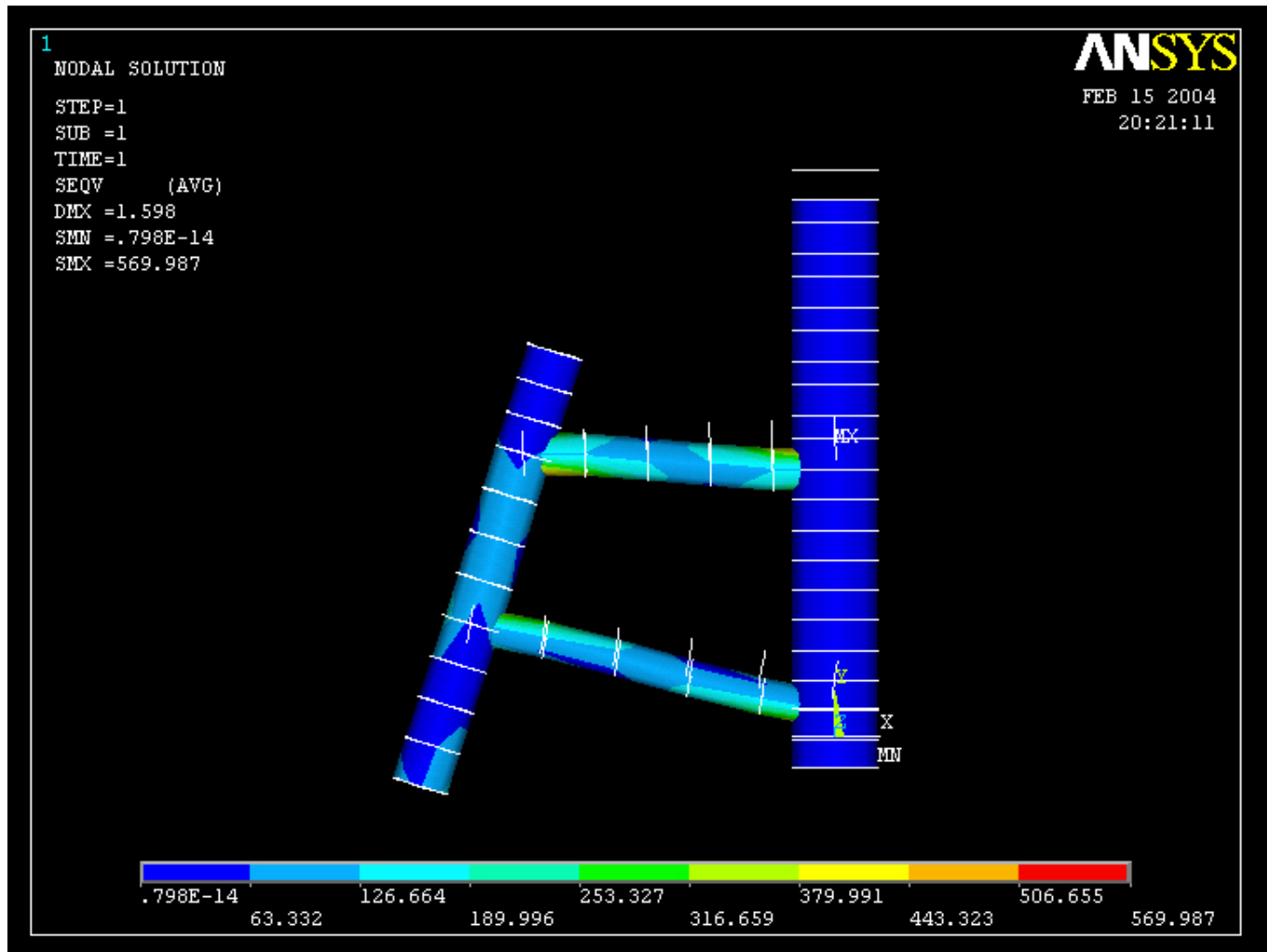
Example – Deformed Shape



Example – Contour Plot



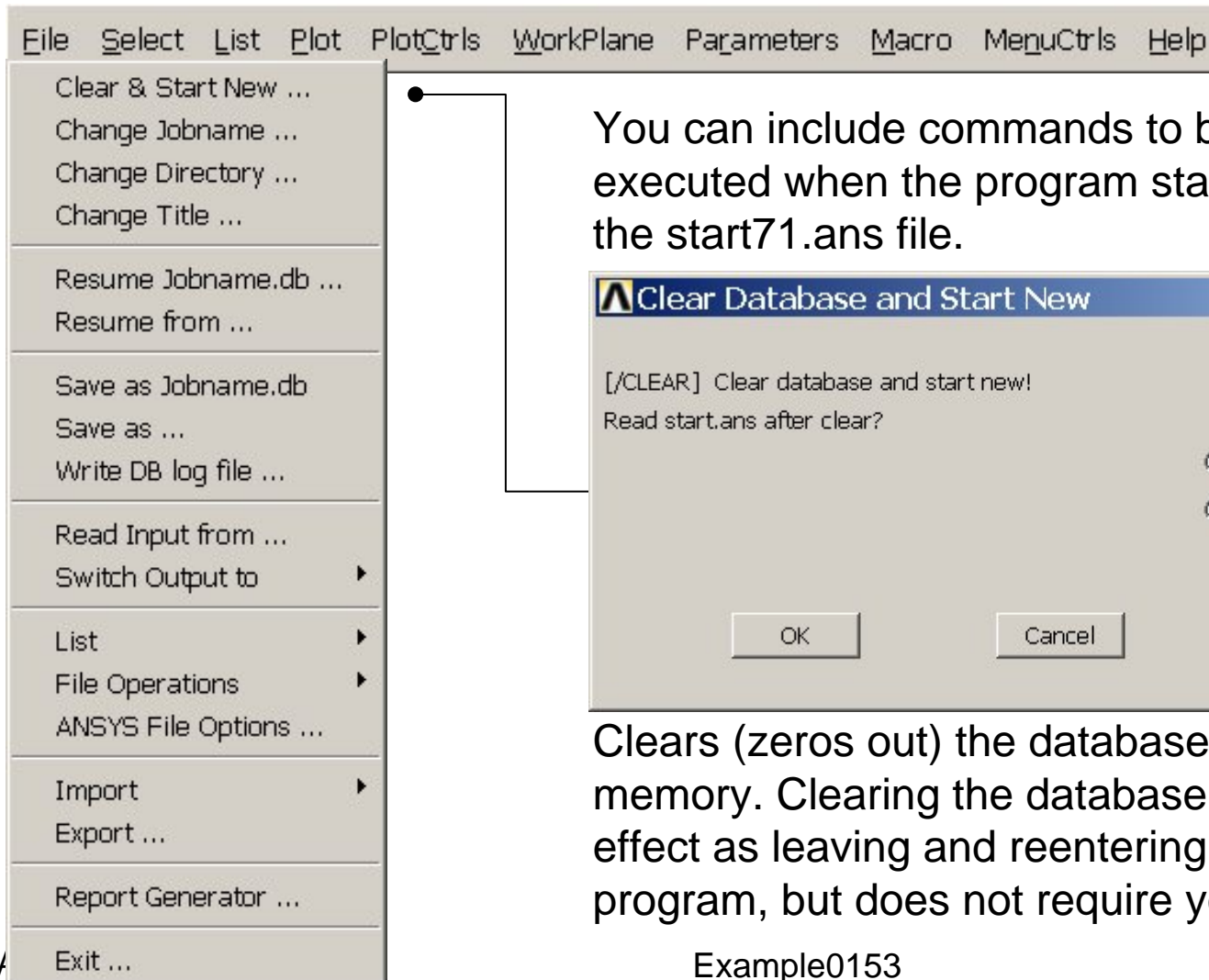
Example – Contour Plot



Example – Comments/Questions

- Could the model be modeled with beam elements instead of pipe elements?
- The “example0153.lgw” can be edited in “Notepad”
- Will the number of elements affect the solution?

File menu



You can include commands to be executed when the program starts up in the start71.ans file.

Clears (zeros out) the database stored in memory. Clearing the database has the same effect as leaving and reentering the ANSYS program, but does not require you to exit.