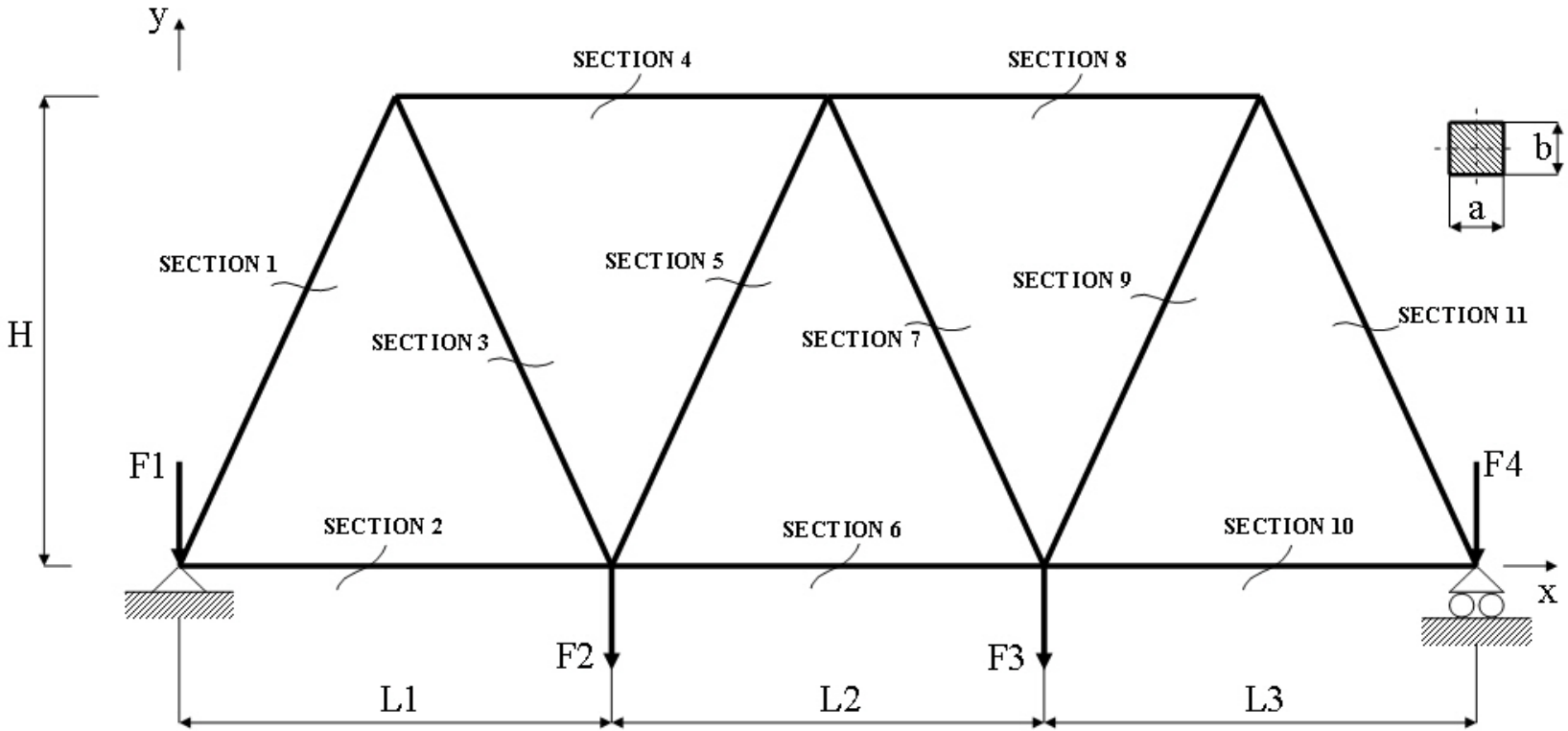
Course in ANSYS

Example0152

Example – Truss 2D



$E = 210e09 N/m^2$

$\nu = 0.3$

$L1 = L2 = L3 = 3.6m$

$H = 3.118m$

$a = b = 0.050mm$

$F1 = 280kN$

$F2 = 210kN$

$F3 = 280kN$

$F4 = 360kN$

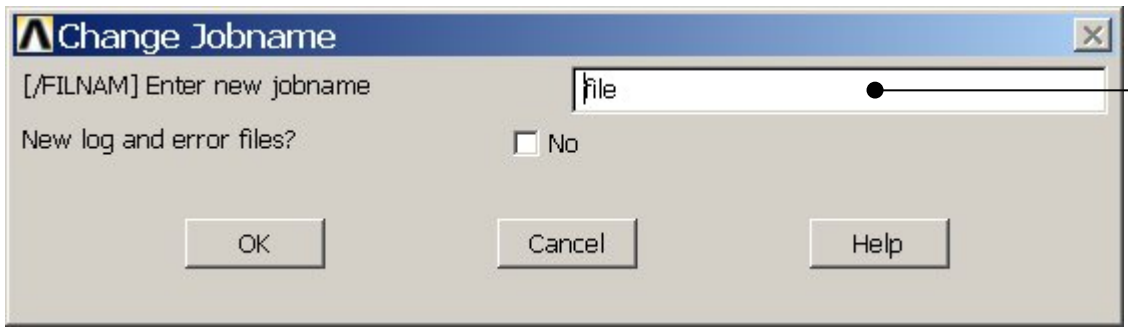
Example – Truss 2D

Objective:
Compute the maximum deflection
Tasks:
Display the deflection figure? Display member forces?
Topics:
Topics: Start of analysis, Element type,
Real constants, Material, modeling, ele-
ment size for beam models, saving/restoring

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0152 — Command line entry

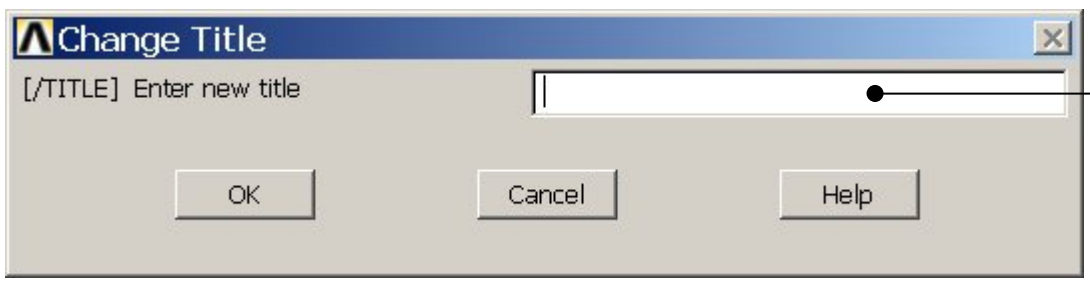


Enter: Example0152

Utility Menu > File > Change Title

/title, Truss 2D

Enter: Truss 2D



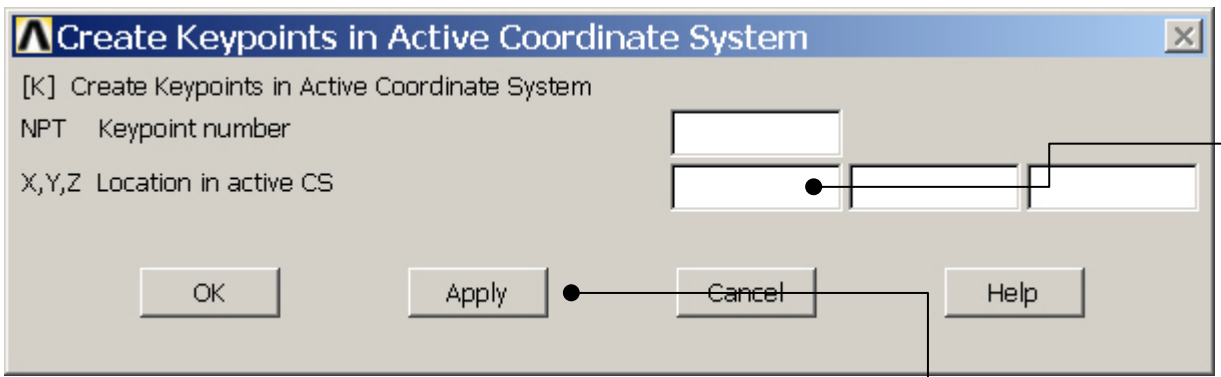
Example - Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS

Note: An empty # result in automatic numbering.

General format:
K,#,X,Y,Z

Keypoint number
X Keypoint x-coordinate
Y Keypoint y-coordinate
Z Keypoint z-coordinate

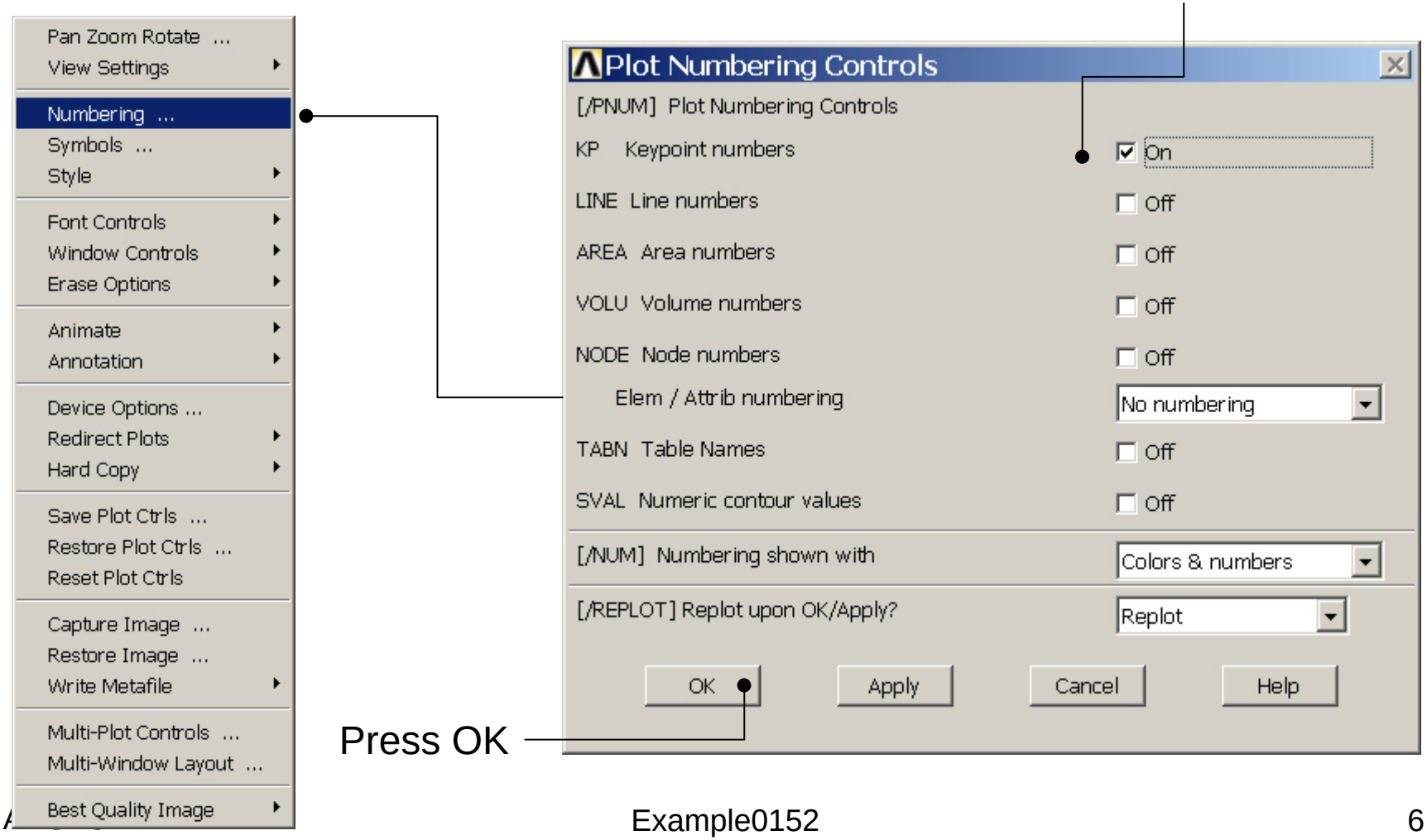


Enter 0,0,0
Enter 3.6,0,0
Enter 7.2,0,0
Enter 10.8,0,0
Enter 9,3.118,0
Enter 4.8, 3.118,0
Enter 1.8, 3.118,0

Example - Numbering

Utility Menu > PlotCtrls > Numbering

Switch on Keypoint numbers

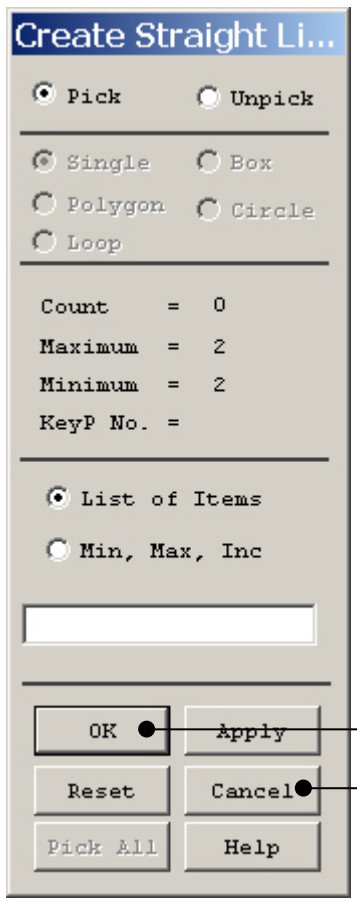


Press OK

Example - Lines

Preprocessor > Modeling > Create > Lines > Lines > Straight Line

L,1,2
L,2,3
L,3,4
L,4,5
L,5,3
L,5,6
L,3,6
L,6,2
L,6,7
L,2,7
L,7,1



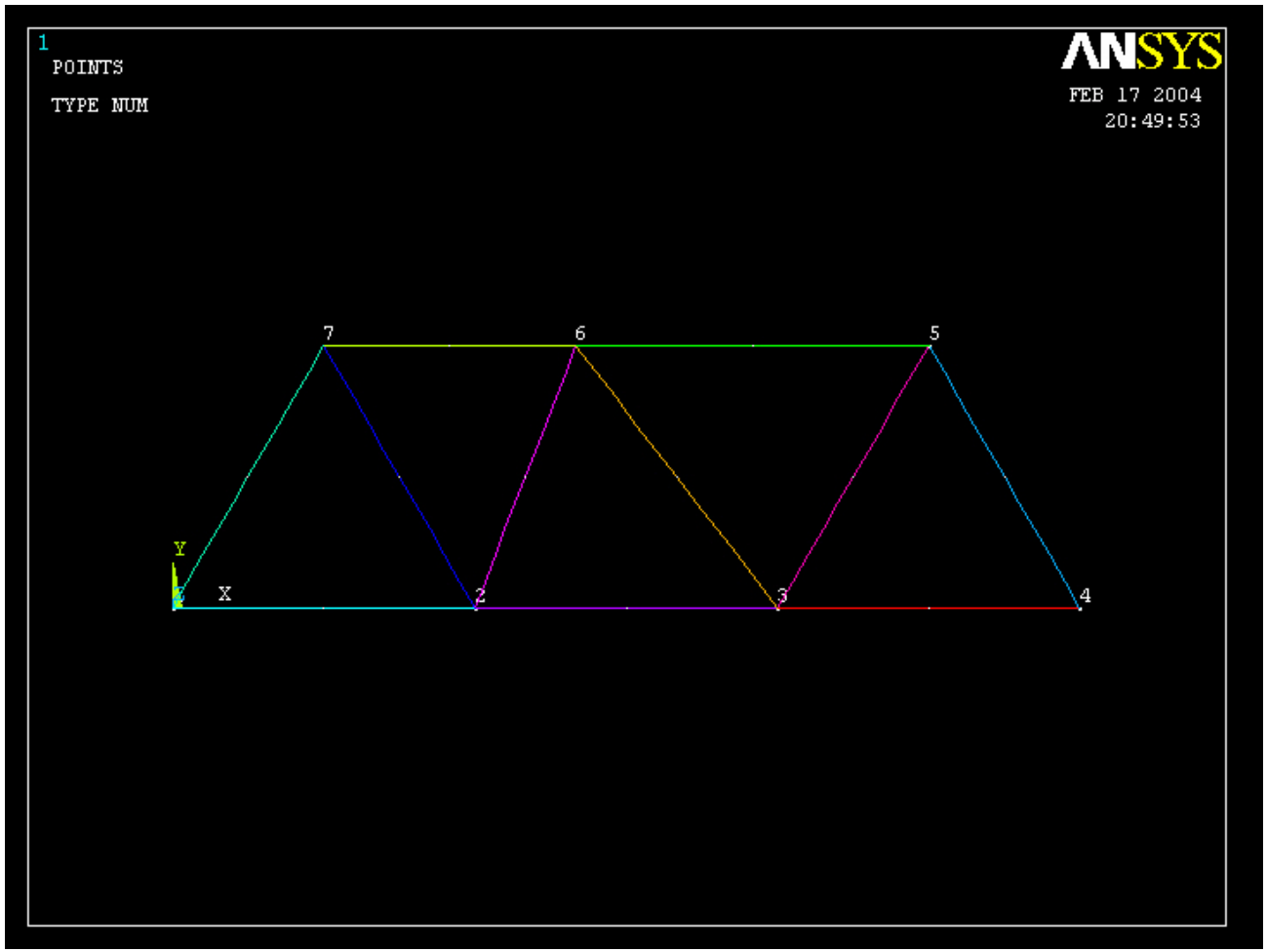
HINT: By clicking with the right-hand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

Pick: upward arrow

Unpick: downward arrow

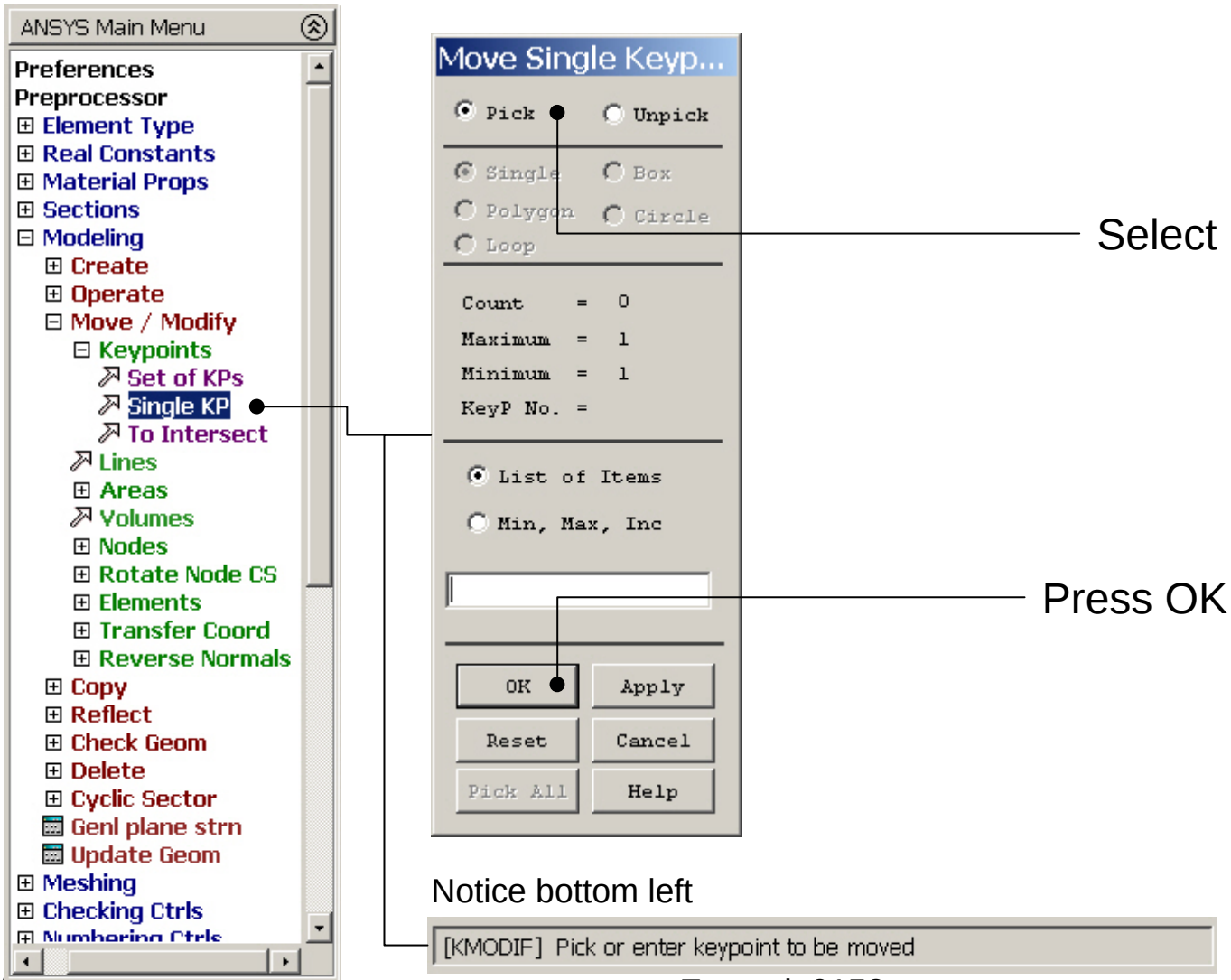
Press OK or Cancel to finish selection

Example – Move/Modify

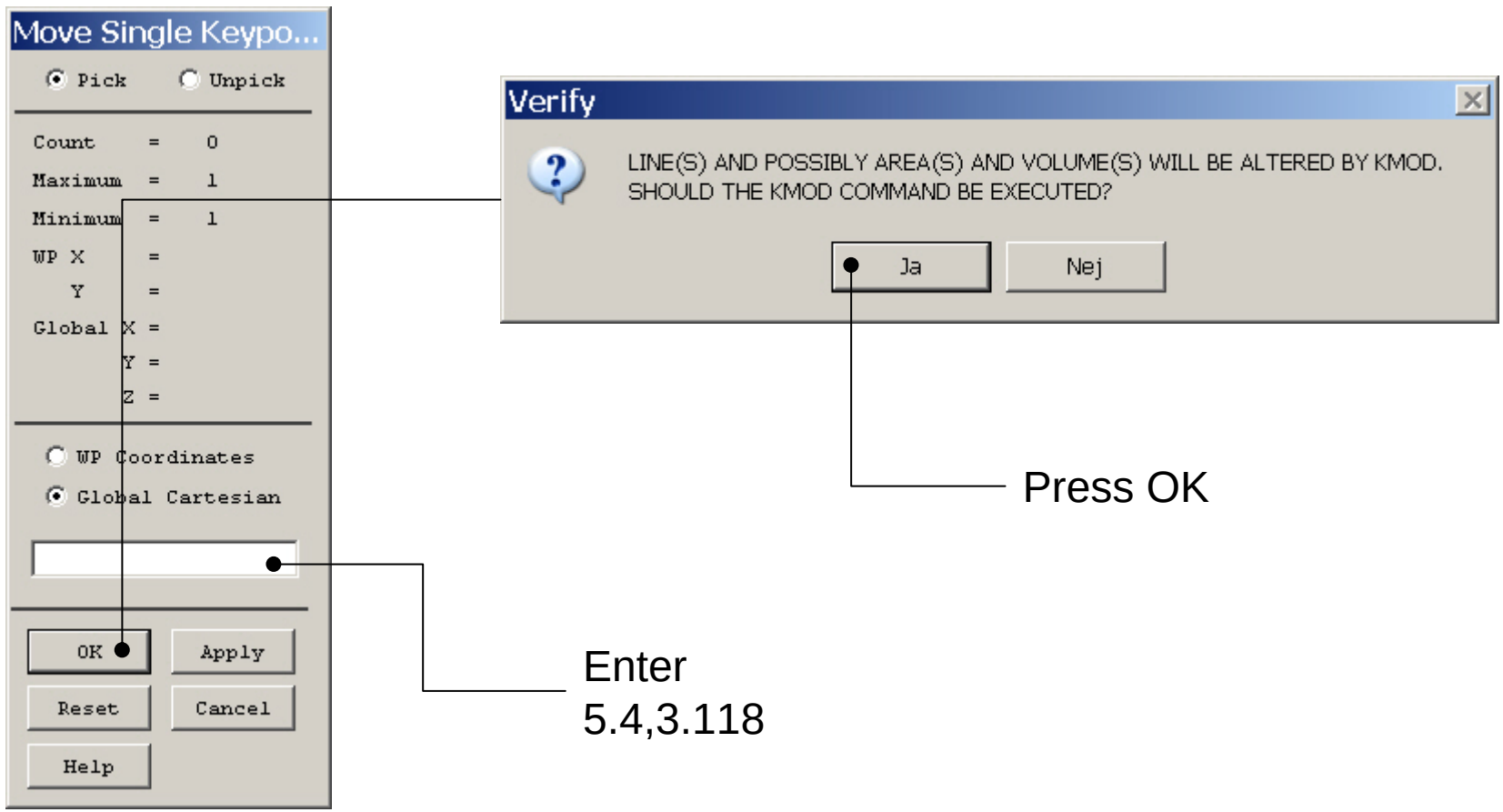


After reconsidering the coordinates it is found that keypoint 6 is errorneous

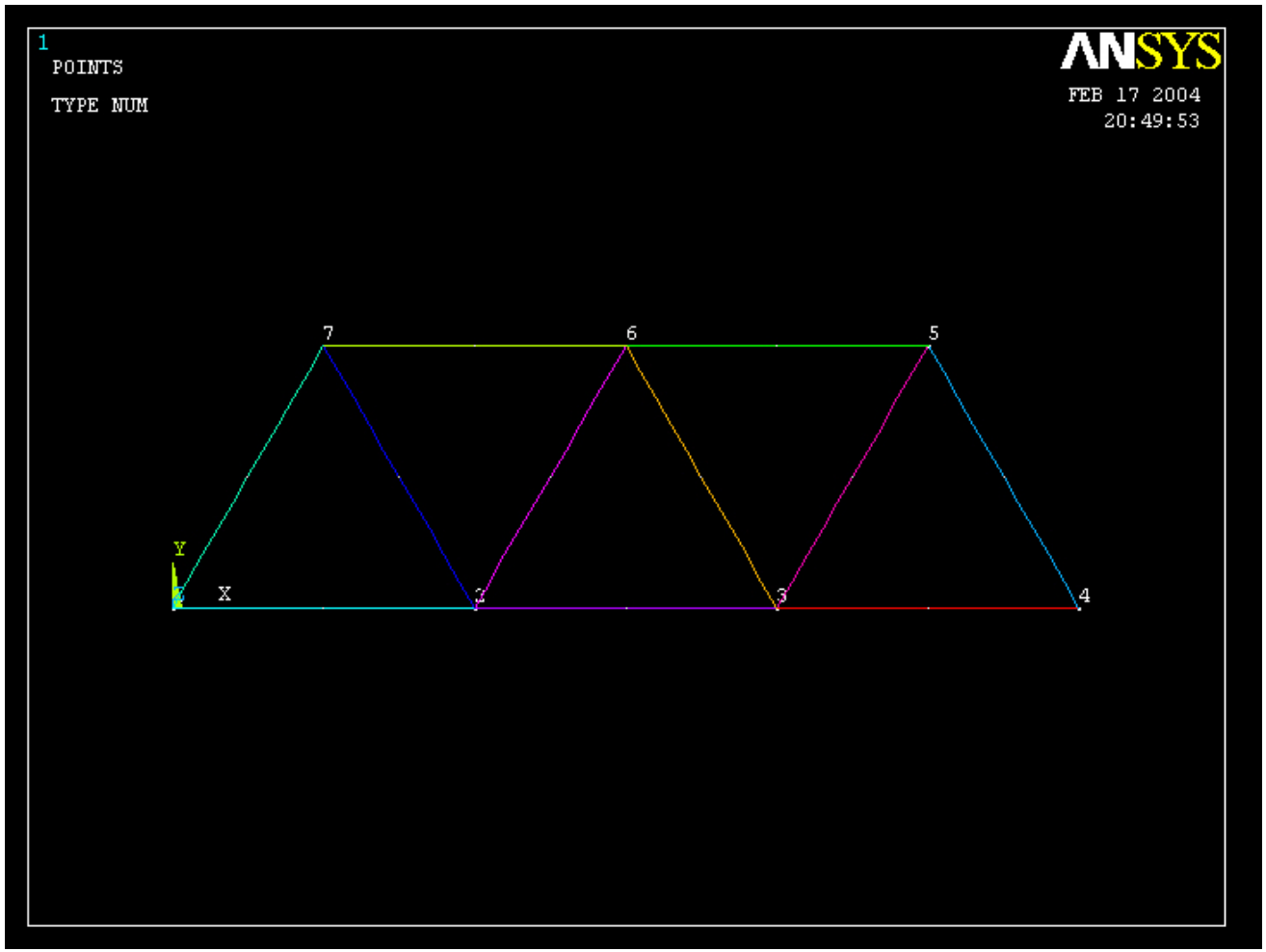
Example – Move Single Keypoint



Example – Move Single Keypoint

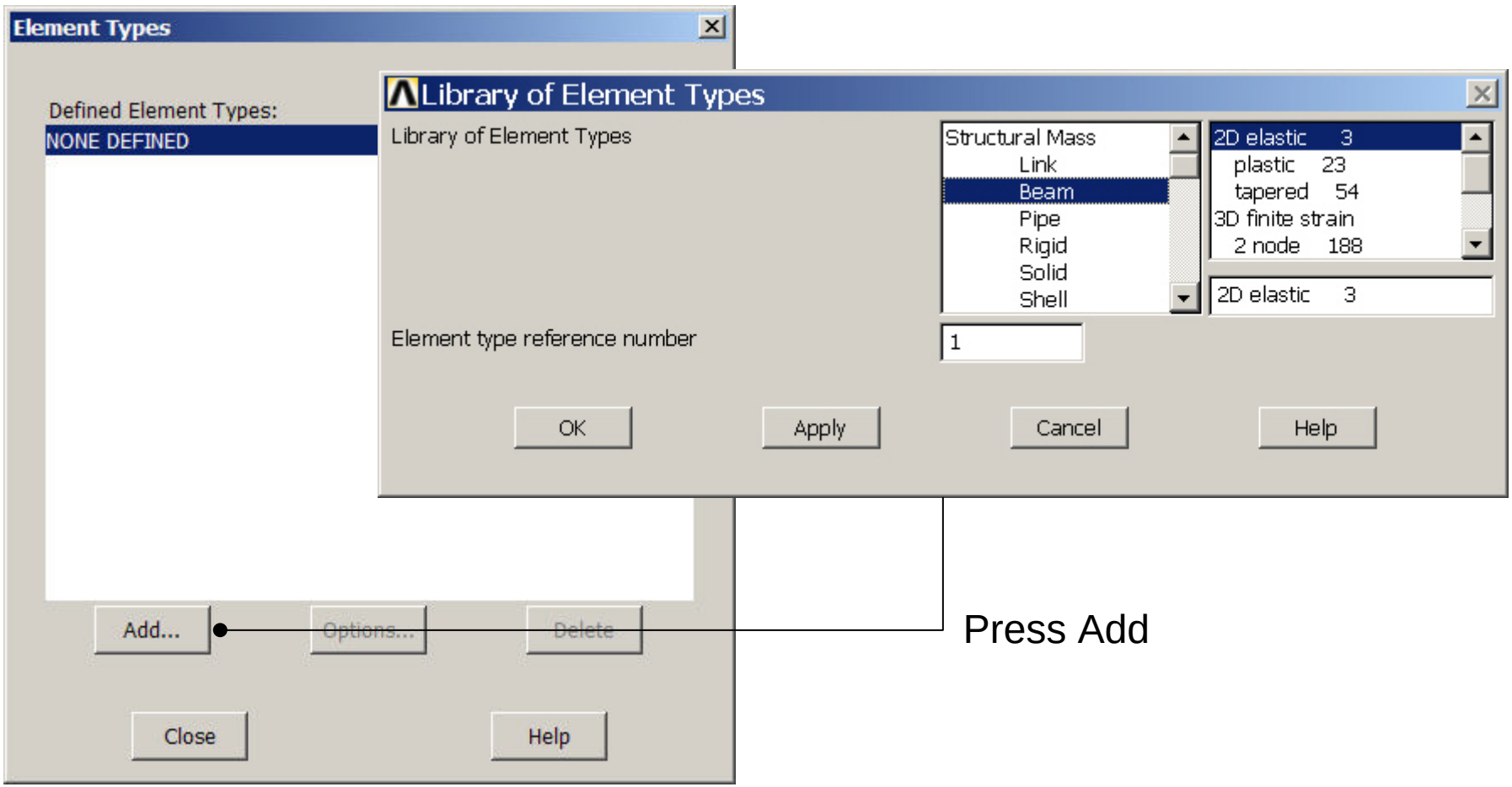


Example – Move Single Keypoint



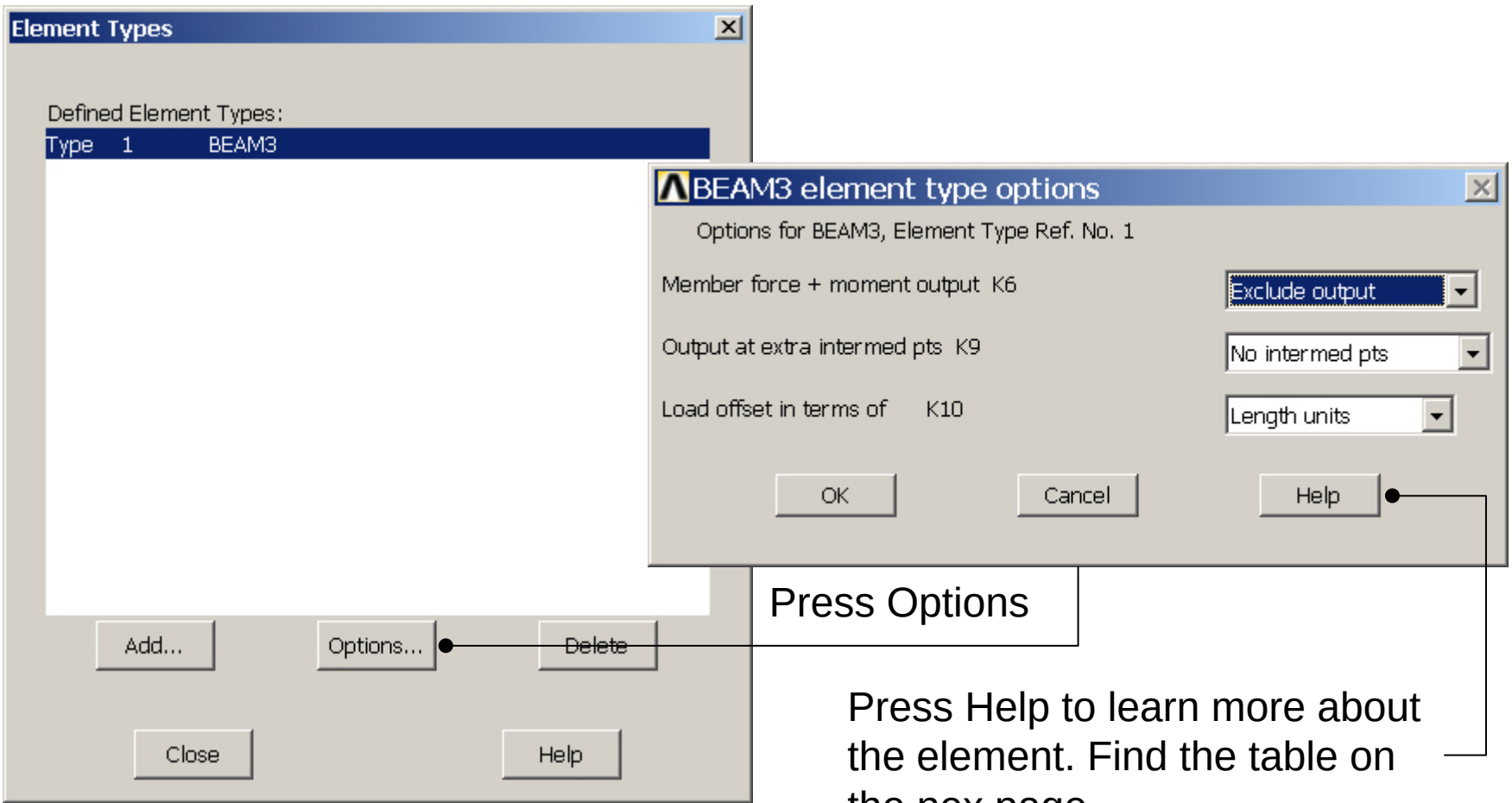
Example – Element Type

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. Find the table on the nex page

Example – Element Table

Find the following table for the element. Identify how to plot member forces in longitudinal direction of the beam element - MFORX

Name	Definition	O	R
EL	Element Number	Y	Y
NODES	Element nodes - I, J	Y	Y
MAT	Element material number	Y	Y
VOLU:	Element volume	N	Y
XC, YC	Location where results are reported	Y	3
TEMP	Temperatures T1, T2, T3, T4	Y	Y
PRES	Pressure P1 at nodes I,J; OFFST1 at I,J; P2 at I,J; OFFST2 at I, J; P3 at I; P4 at J	Y	Y
SDIR	Axial direct stress	1	1
SBYT	Bending stress on the element +Y side of the beam	1	1
SBYB	Bending stress on the element -Y side of the beam	1	1
SMAX	Maximum stress (direct stress + bending stress)	1	1
SMIN	Minimum stress (direct stress - bending stress)	1	1
EPELDIR	Axial elastic strain at the end	1	1
EPELBYT	Bending elastic strain on the element +Y side of the beam	1	1
EPELBYB	Bending elastic strain on the element -Y side of the beam	1	1
EPTHDIR	Axial thermal strain at the end	1	1
EPTHBYT	Bending thermal strain on the element +Y side of the beam	1	1
EPTHBYB	Bending thermal strain on the element -Y side of the beam	1	1
EPINAXL	Initial axial strain in the element	1	1
MFOR(X, Y)	Member forces in the element coordinate system X and Y direction	2	Y
MMOMZ	Member moment in the element coordinate system Z direction	2	Y

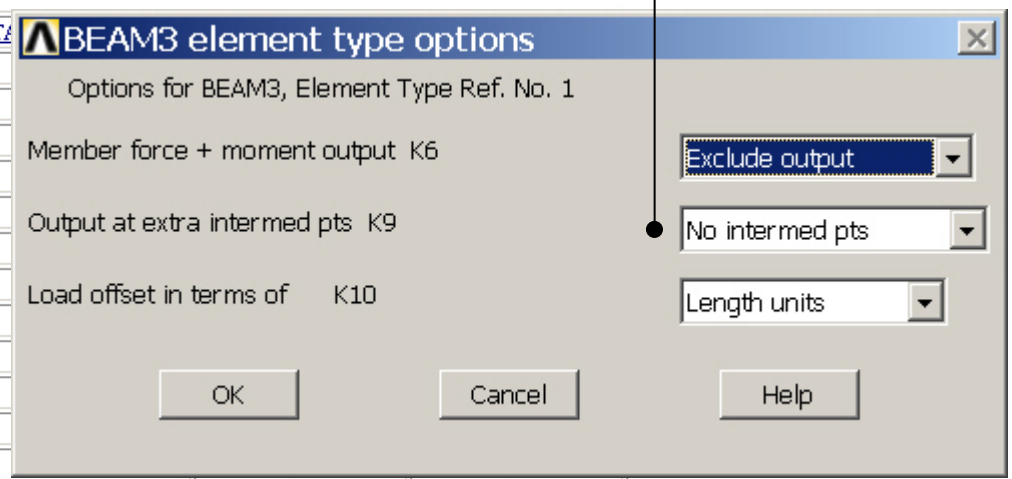
Example – Element Table

Find also the following table in the Help function

Table 3.2. BEAM3 Item and Sequence Numbers (KEYOPT(9) = 0)

Output Quantity Name	Item	ETA		
SDIR	LS			
SBYT	LS			
SBYB	LS			
EPELDIR	LEPEL			
EPELBYT	LEPEL			
EPELBYB	LEPEL			
EPTHDIR	LEPTH			
EPTHBYT	LEPTH			
EPTHBYB	LEPTH			
EPINAXL	LEPTH			
SMAX	NMISC			
SMIN	NMISC	-	2	4
MFORX	SMISC	-	1	7
MFORY	SMISC	-	2	8
MMOMZ	SMISC	-	6	12
P1	SMISC	-	13	14
OFFST1	SMISC	-		
P2	SMISC	-		
OFFST2	SMISC	-	19	20
P3	SMISC	-	21	-
P4	SMISC	-	-	22

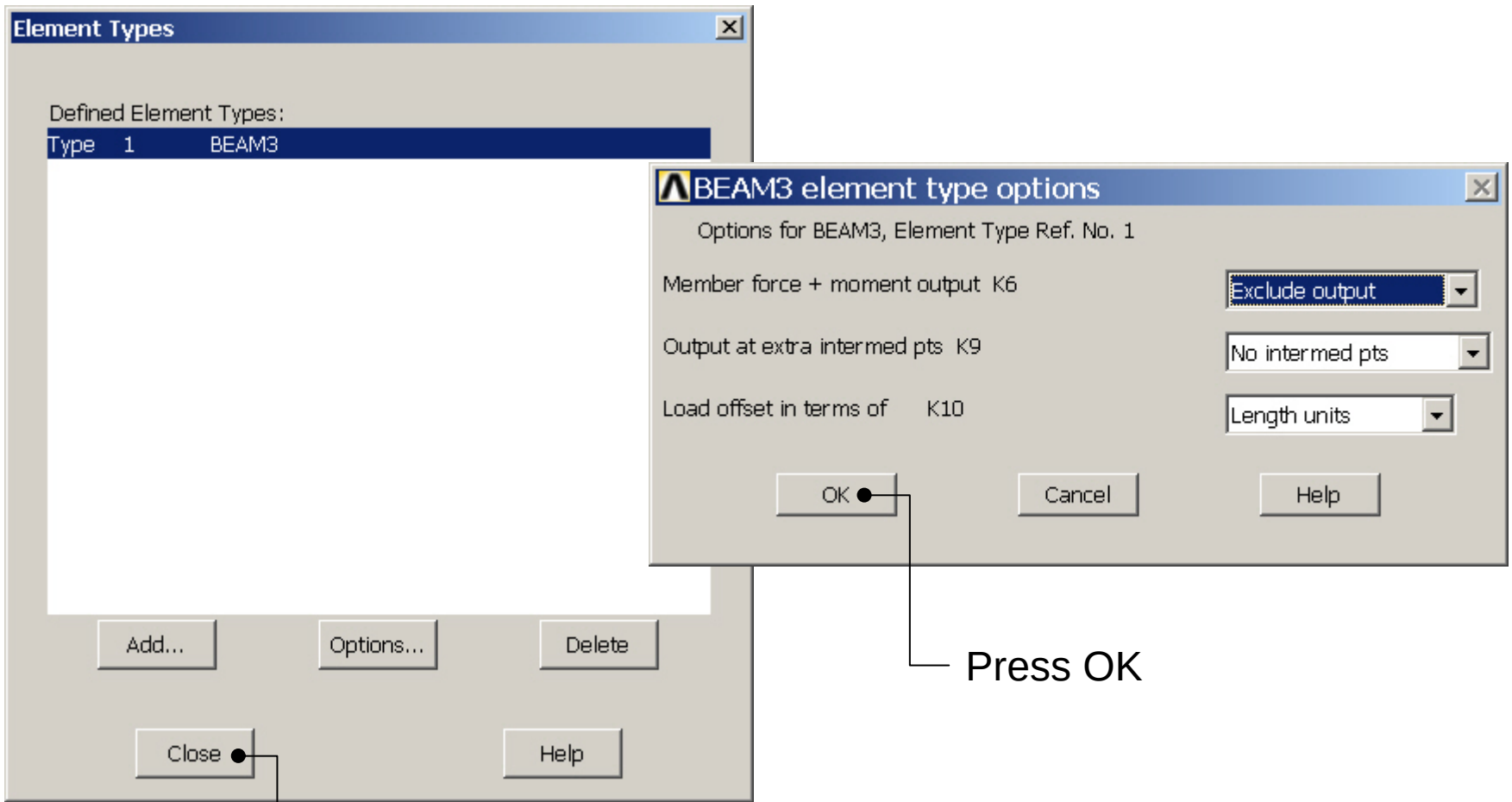
		Pseudo Node			
		1	2	3	4
TEMP	LBFE	1	2	3	4



Remember MFORX, SMISC,1,7

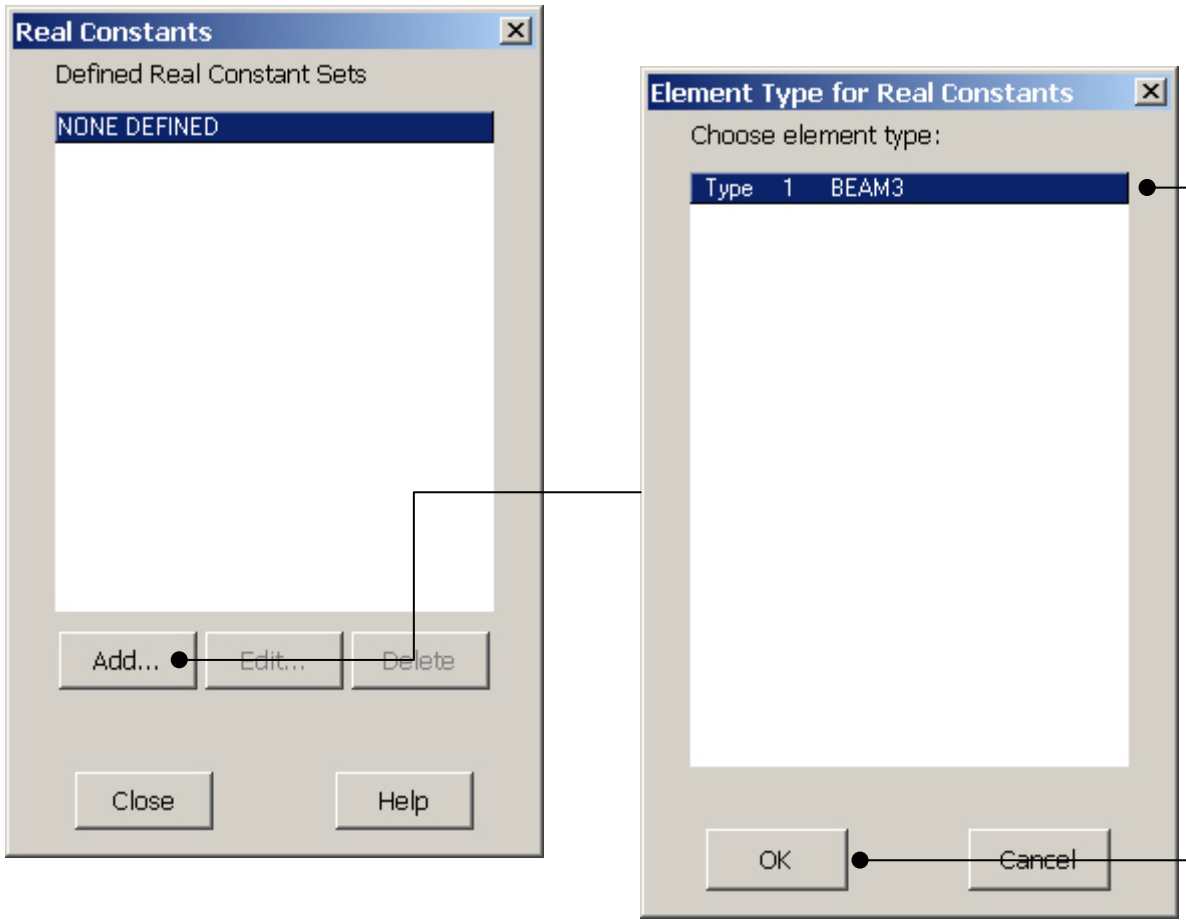
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Real Constants

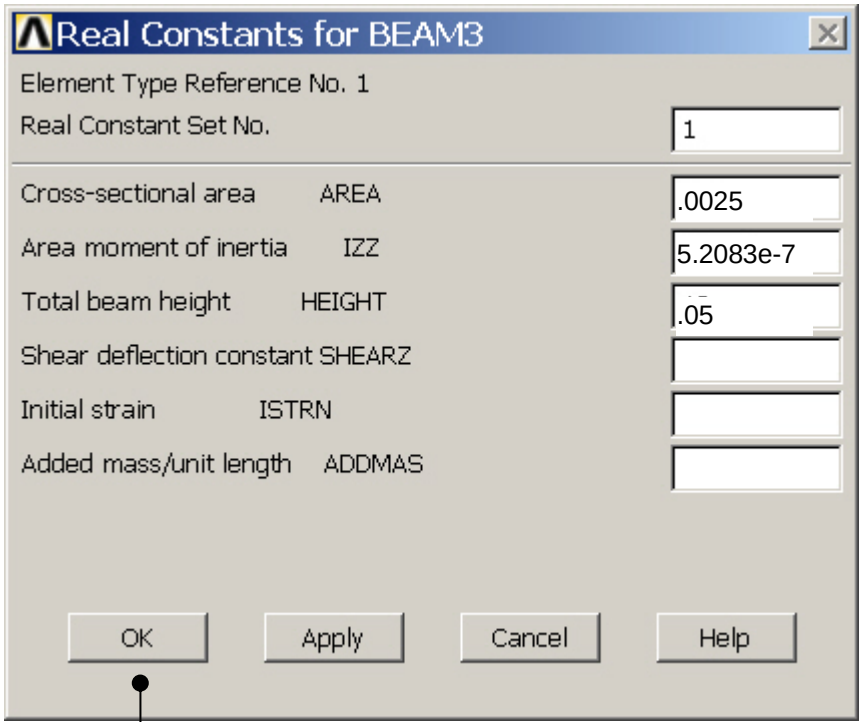
Preprocessor > Real Constants > Add

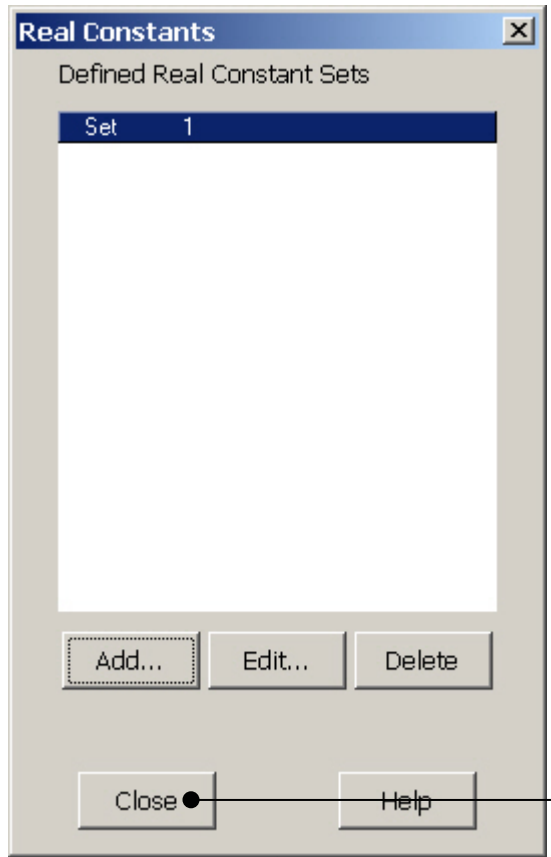


Place the cursor on the relevant element and press OK

Example - Real Constants

Preprocessor > Real Constants > Add



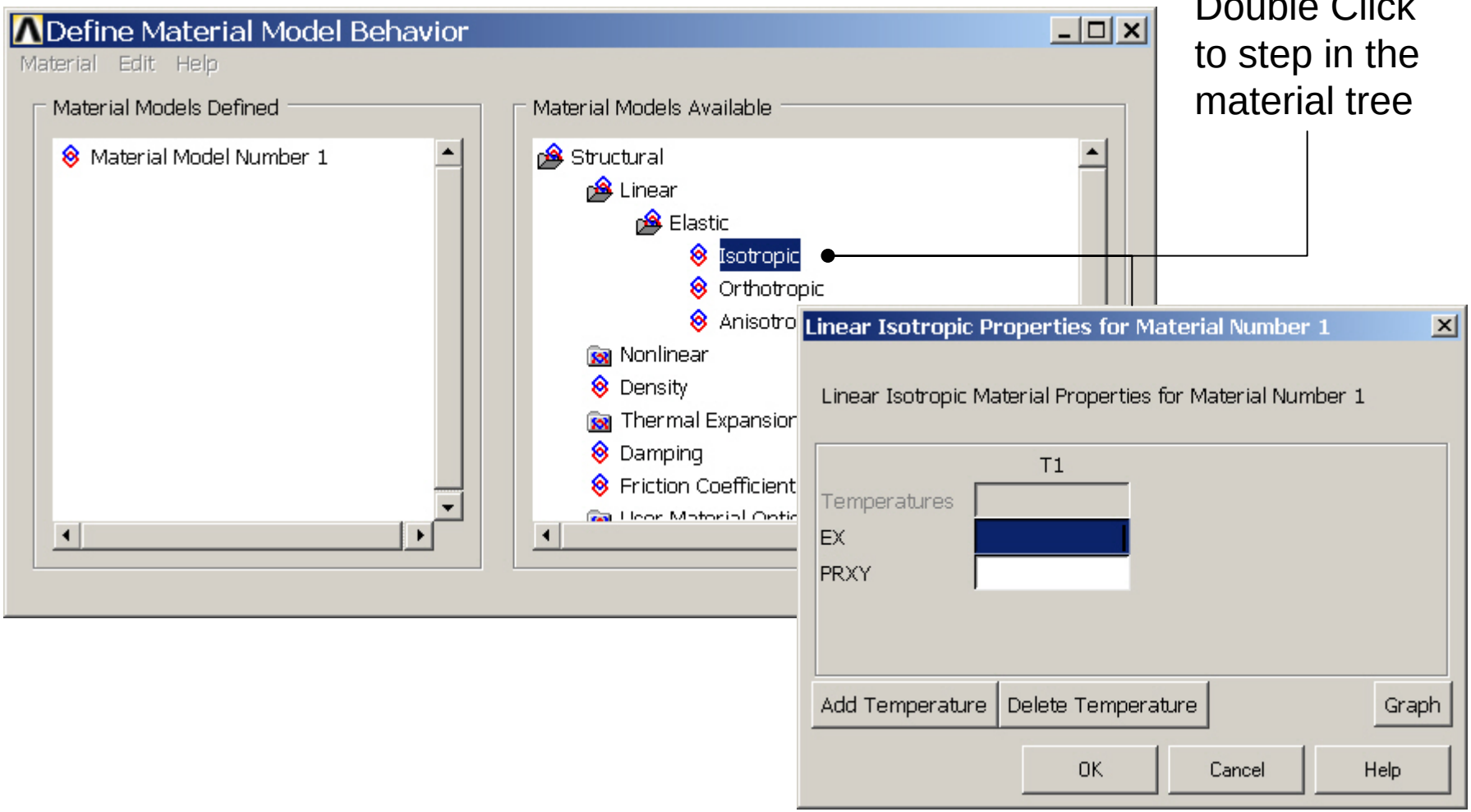


Press Close to finish

Press OK

Example - Material Properties

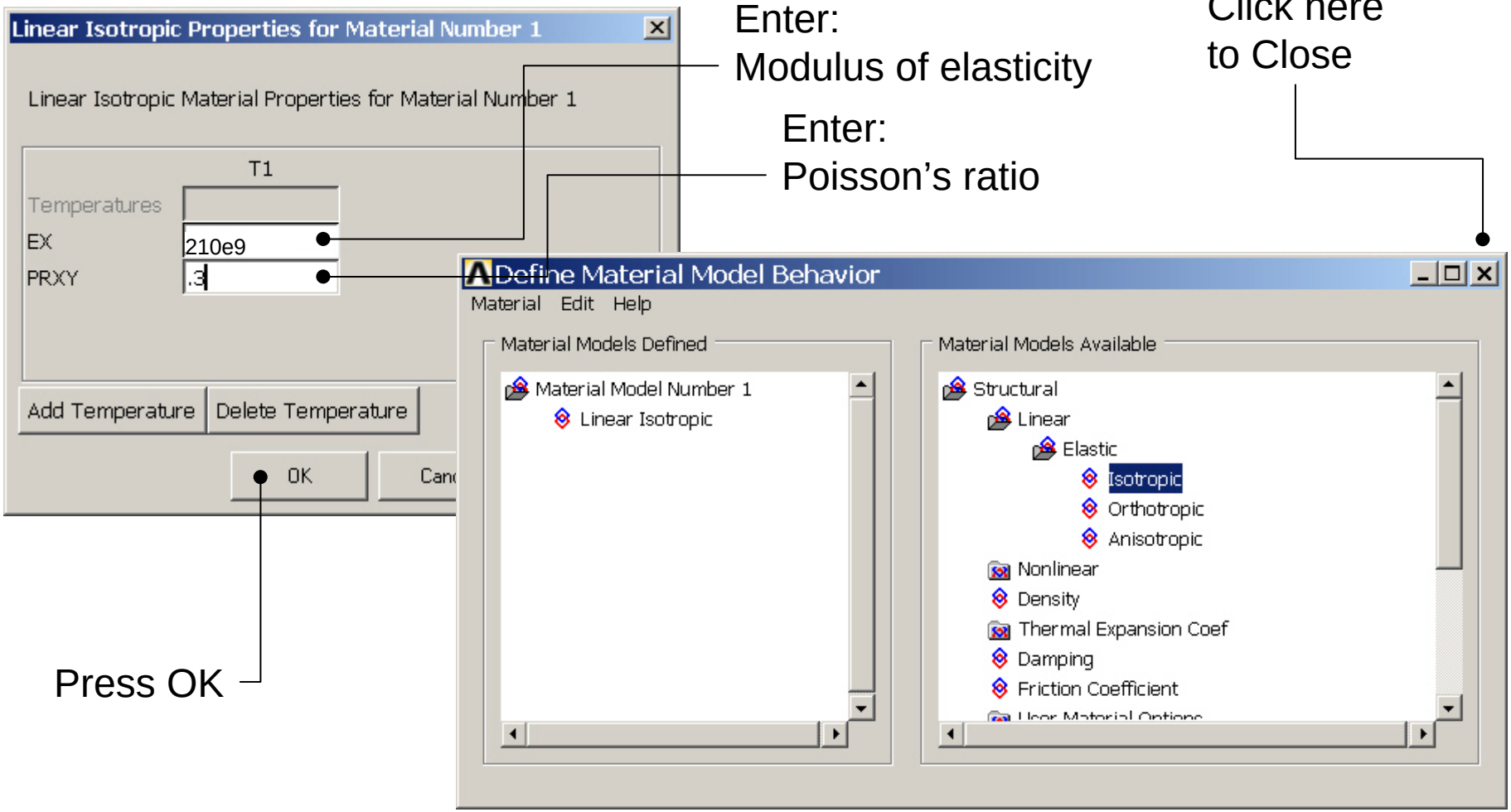
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

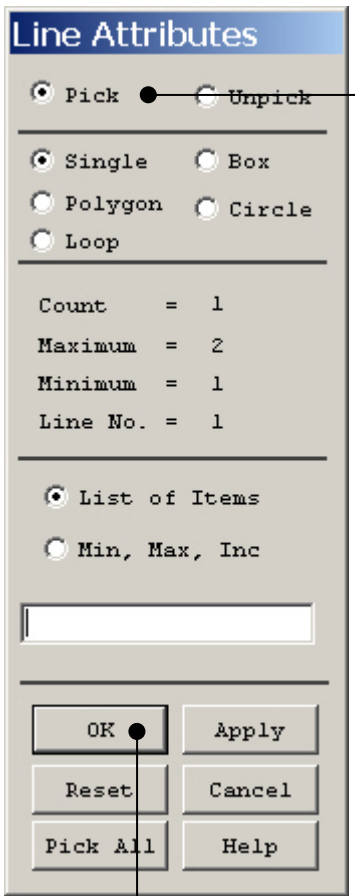
Example - Material Properties

Preprocessor > Material Props > Material Models

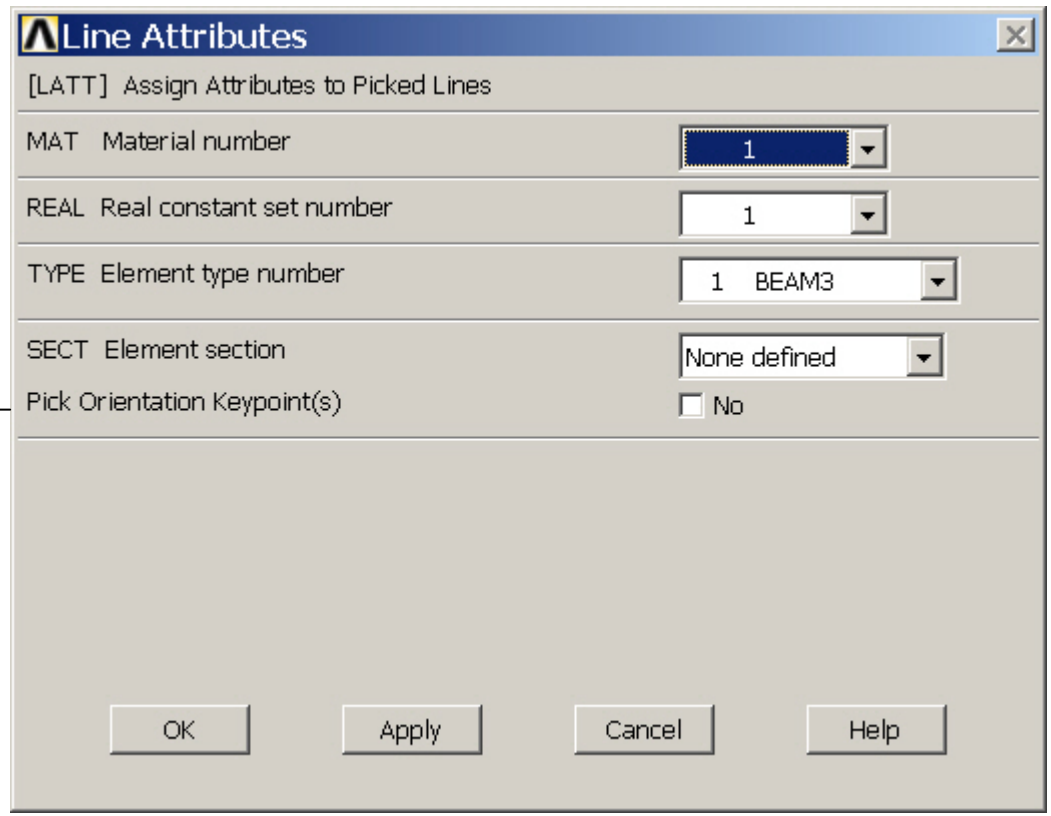


Example – Mesh Attributes

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



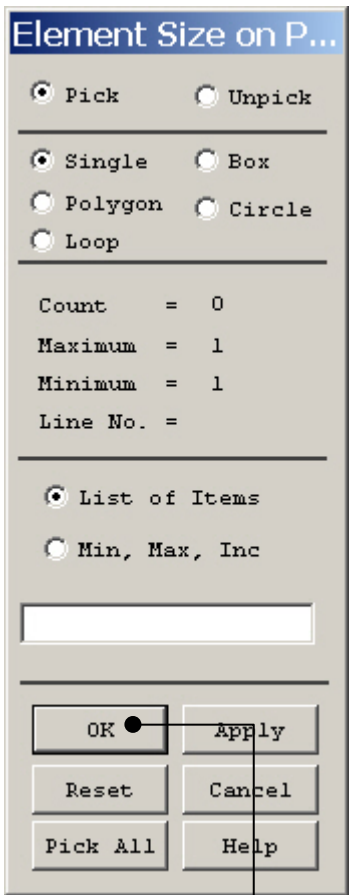
Select All Lines

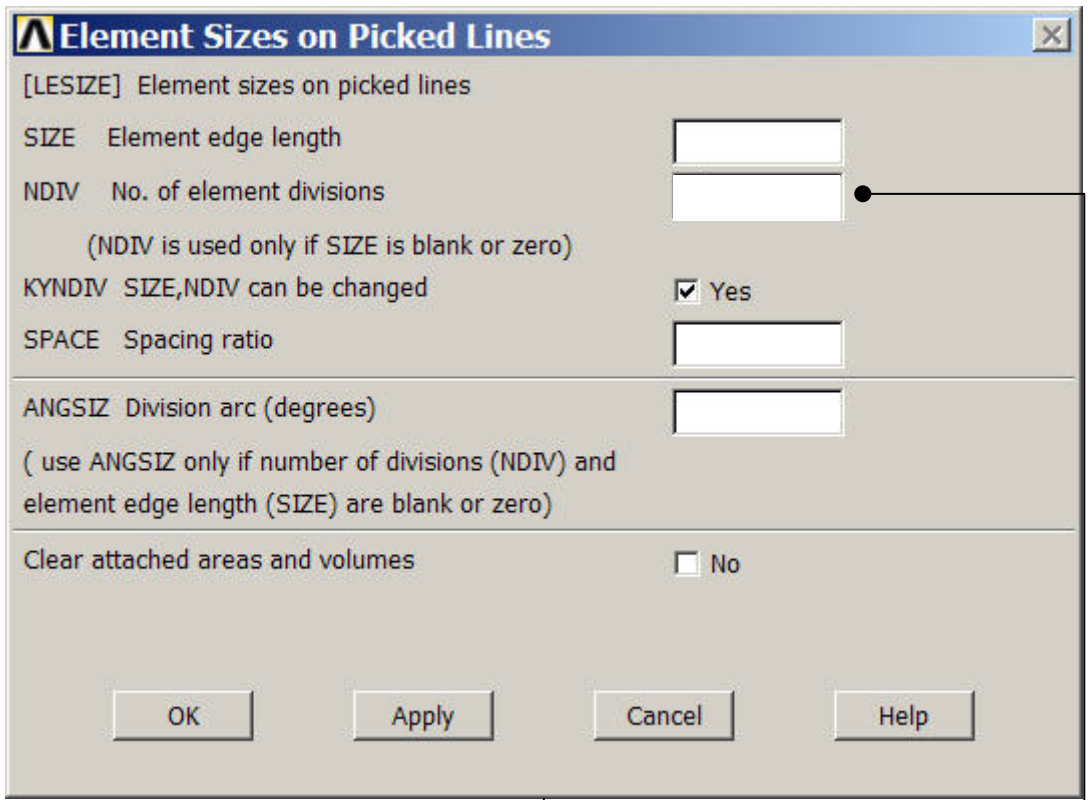


Press OK

Example - Meshing

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





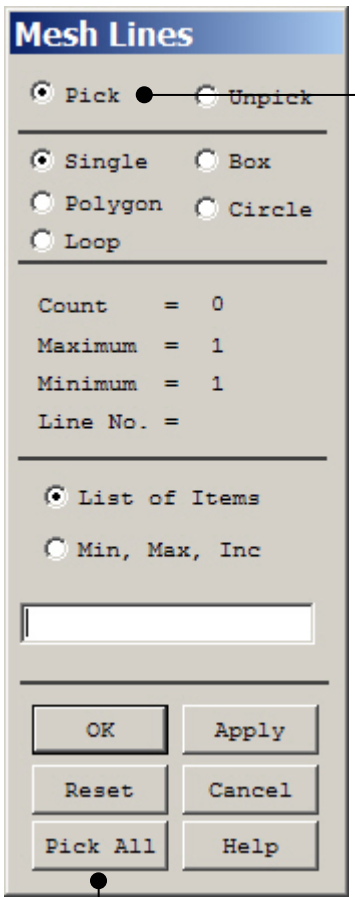
Select/Pick Lines to specify mesh size for – Select All lines

Press OK when finish with selection

Enter 1

Example - Meshing

Preprocessor > Meshing > Mesh > Lines



Select individual lines to be meshed by Picking

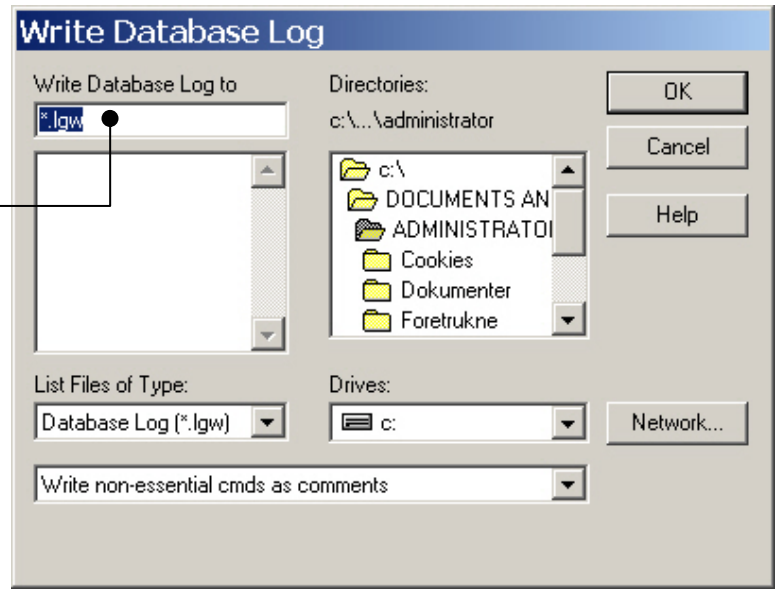
NB: It is often necessary to “Clear” the model for example if Element Type is to be changed

Select all lines defined to be meshed

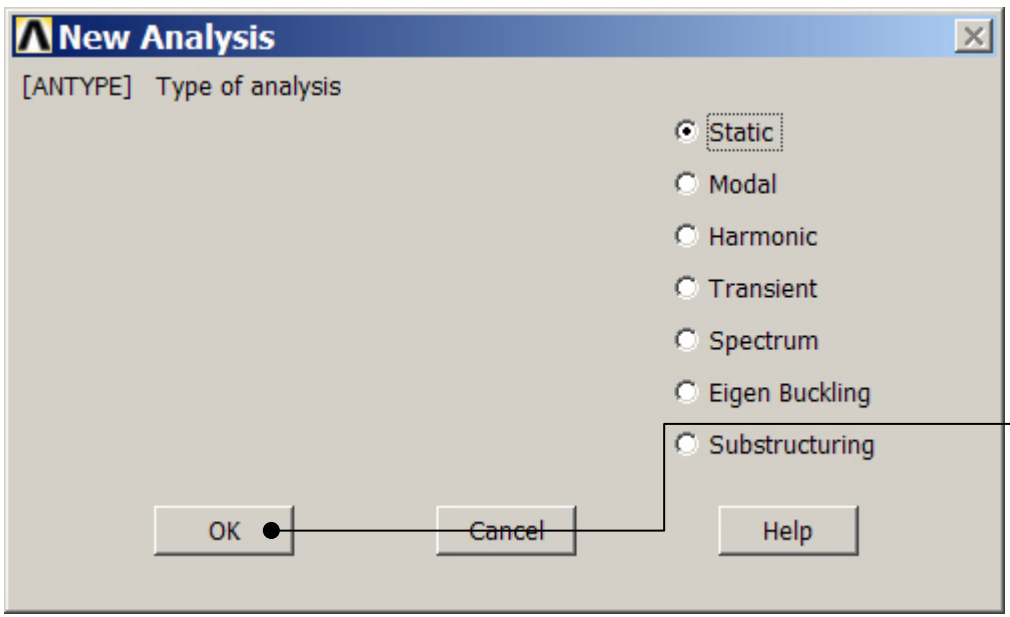
Example – Analysis Type

File > Write DB log file

Enter “example0152.lgw”



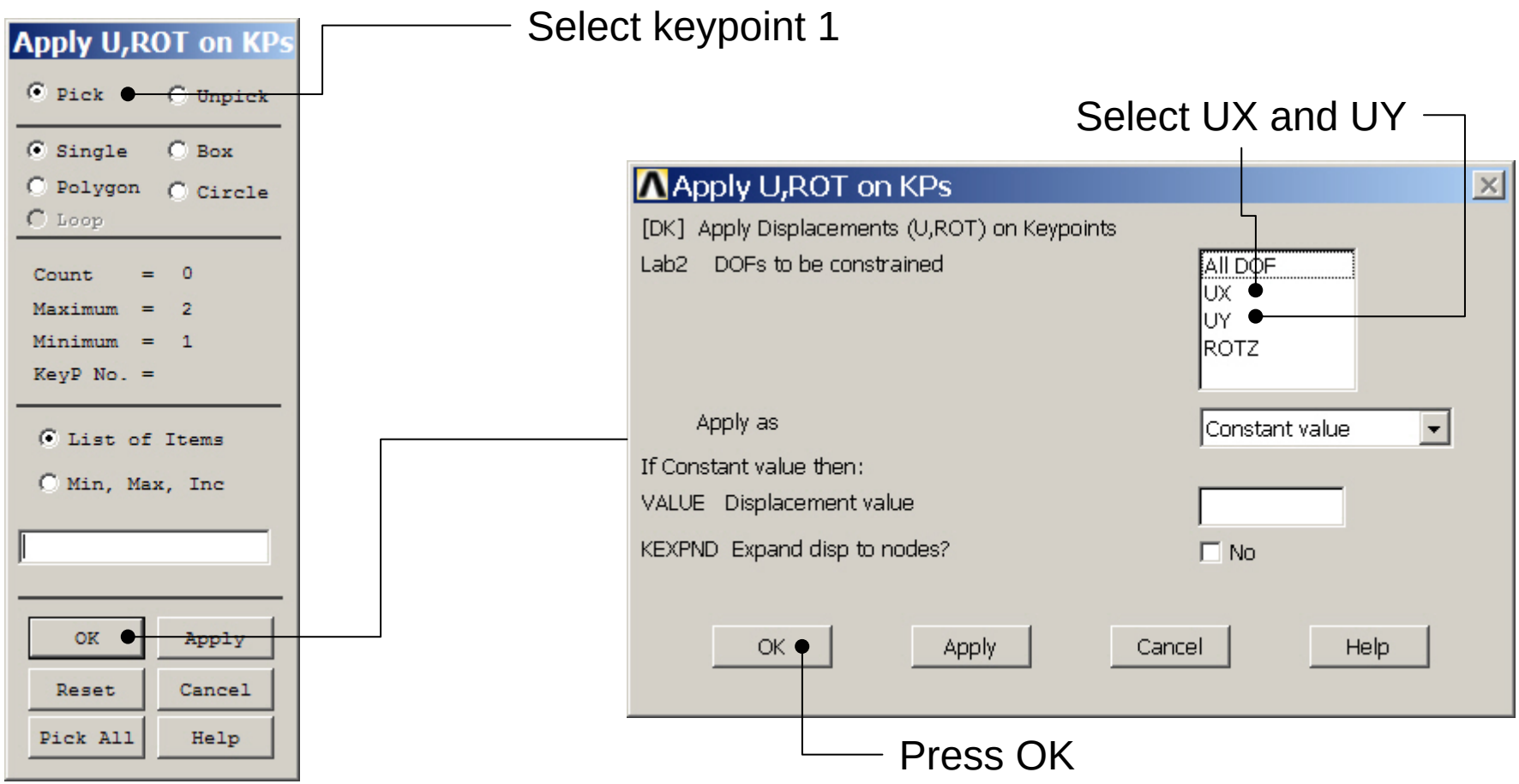
Solution > Analysis Type > New Analysis



Press OK

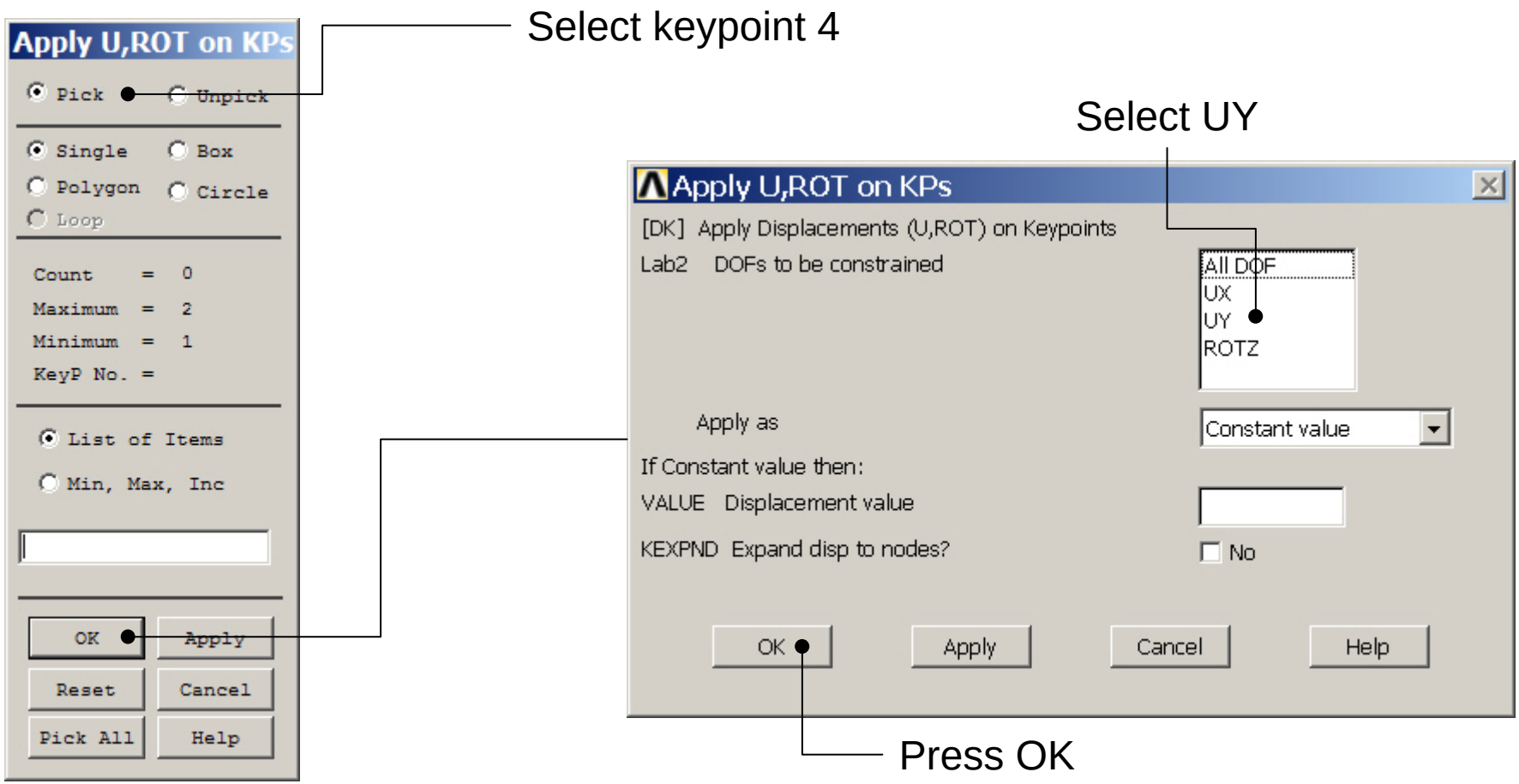
Example – Define Loads

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



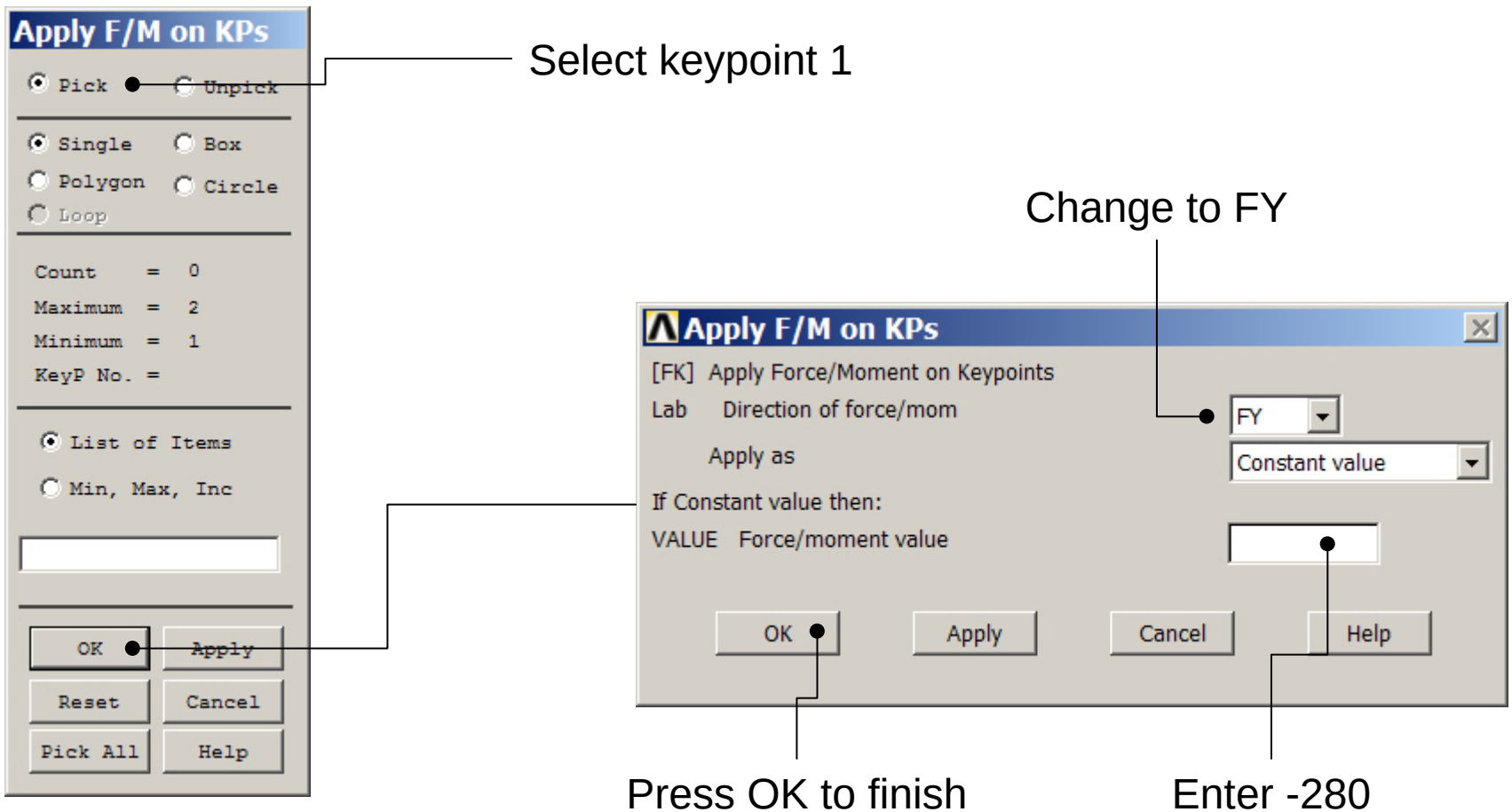
Example – Define Loads

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



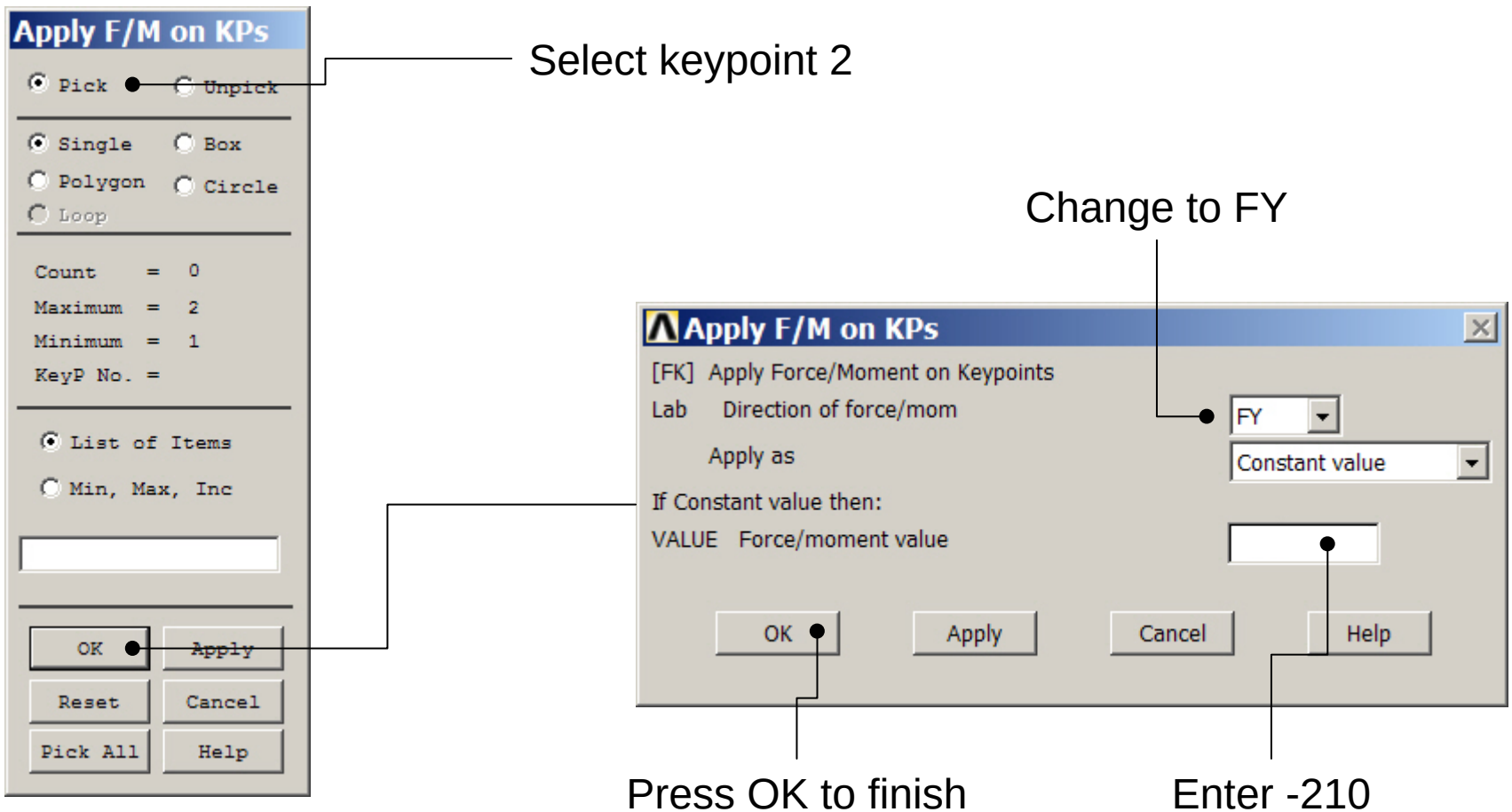
Example – Define Loads

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



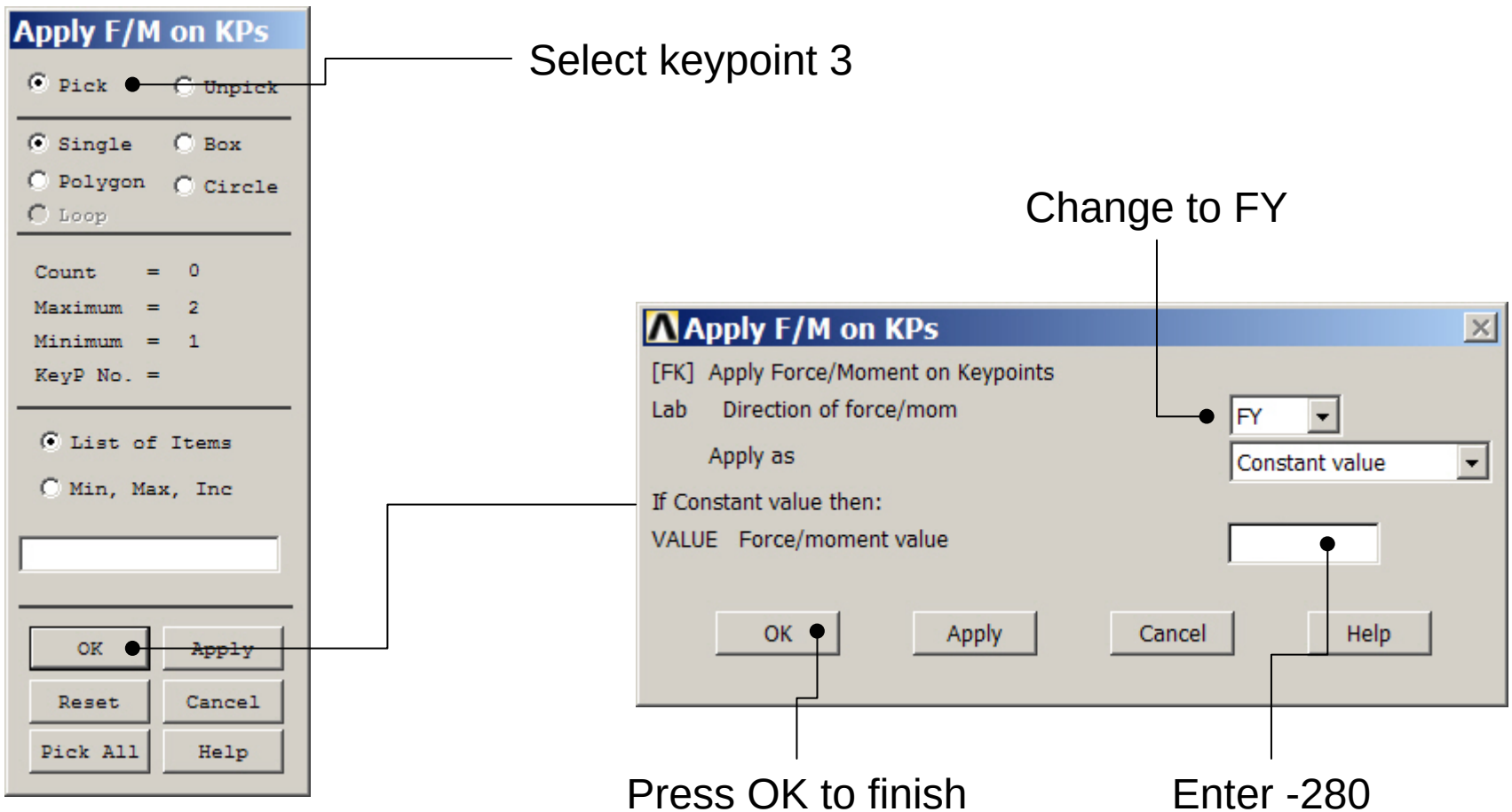
Example – Define Loads

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



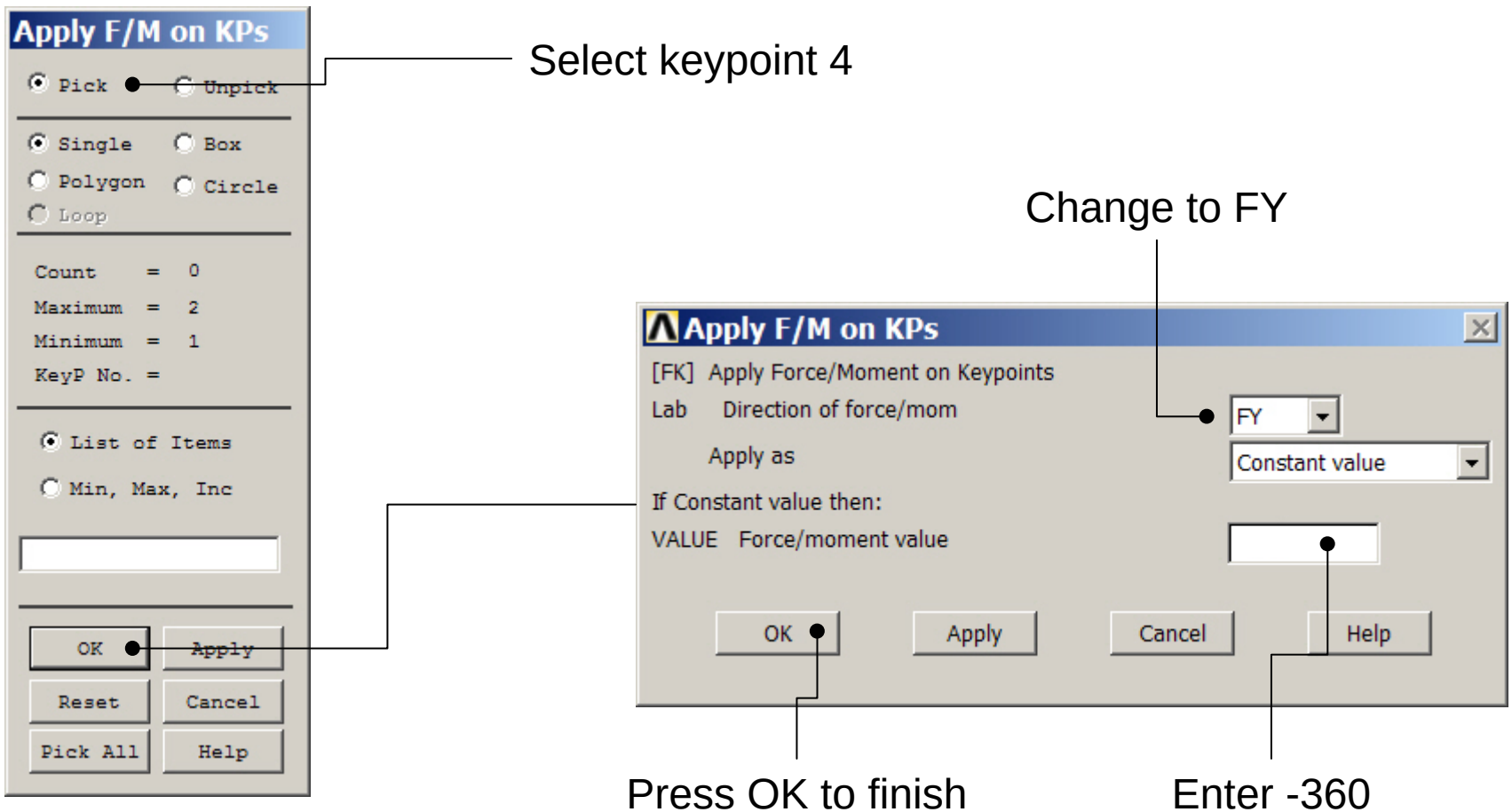
Example – Define Loads

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

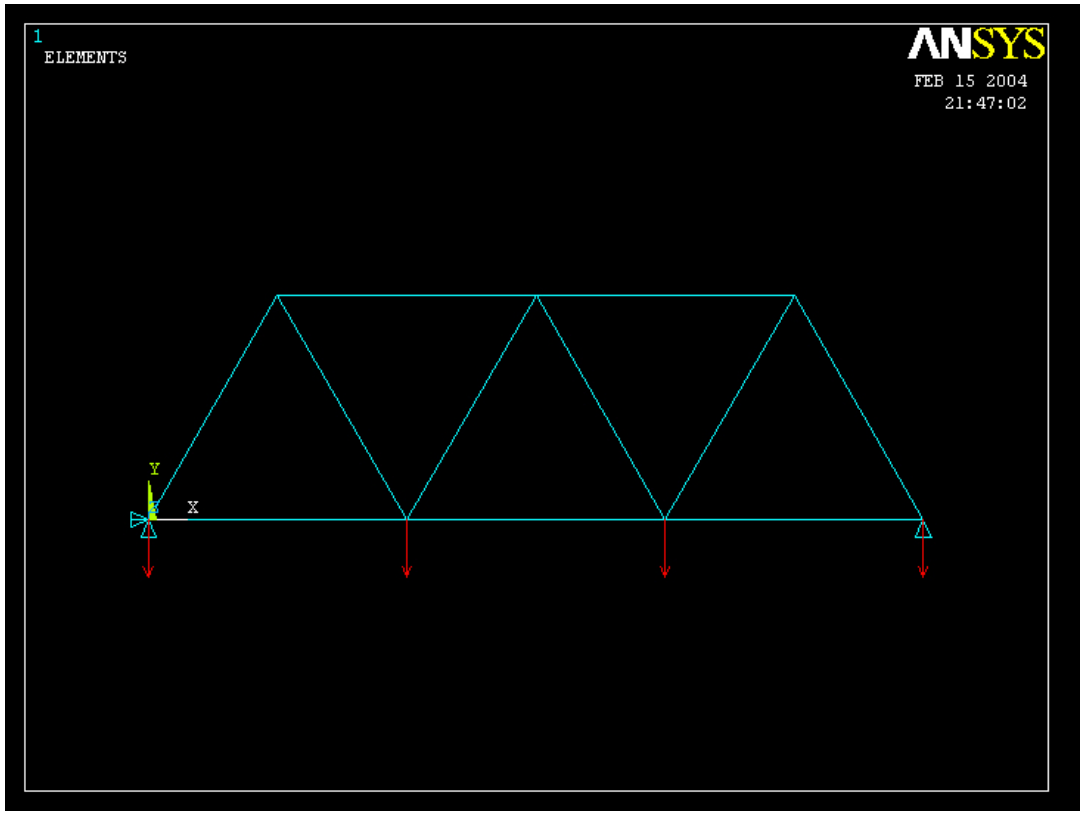


Example – Define Loads

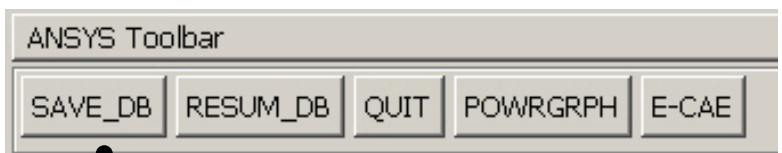
Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints



Example - Save



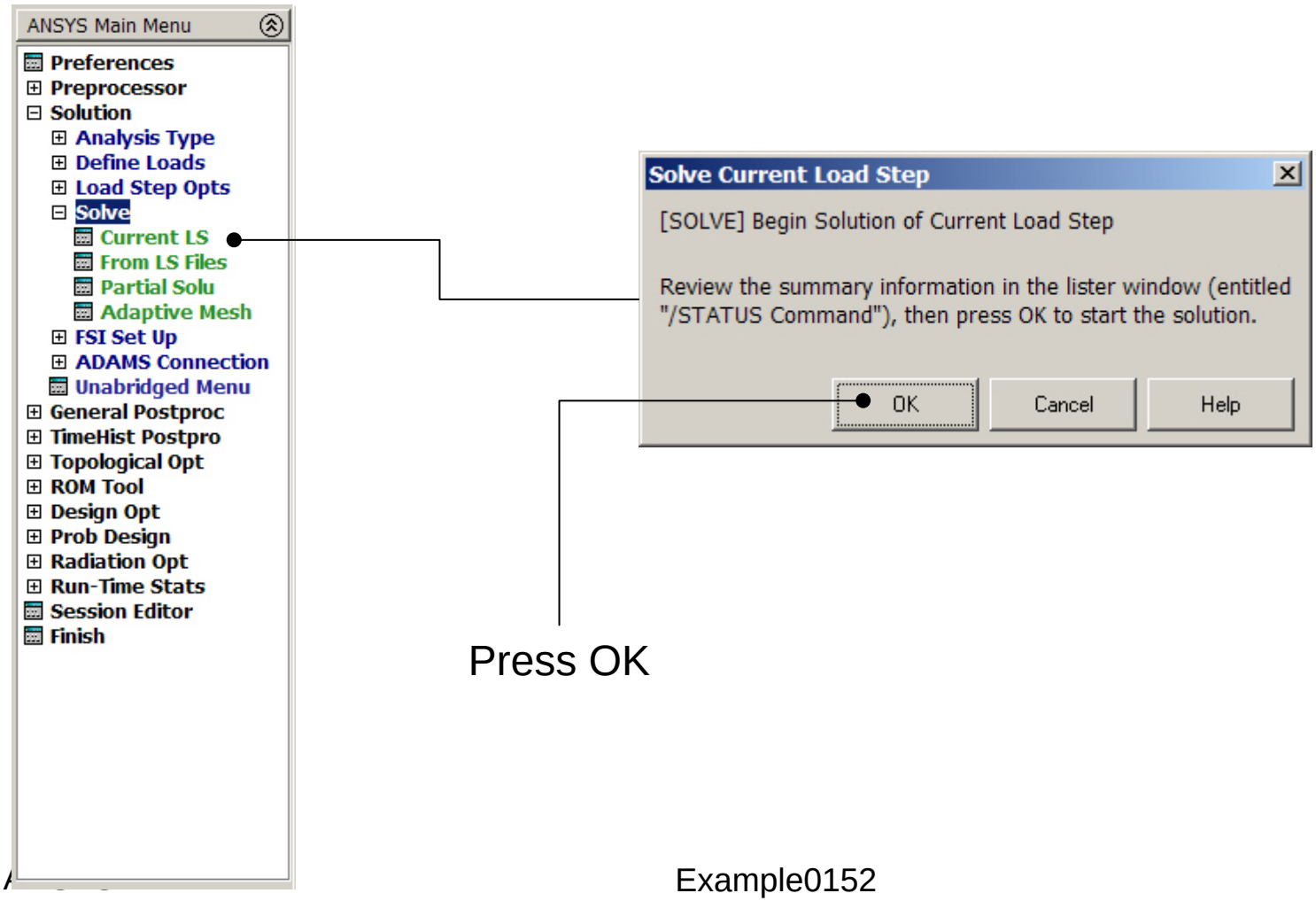
Display of Analysis model



Save the model

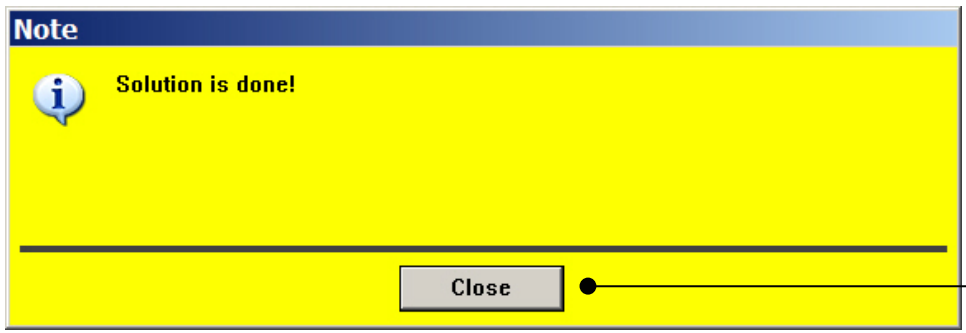
Example - Solve

Solution > Solve > Current LS



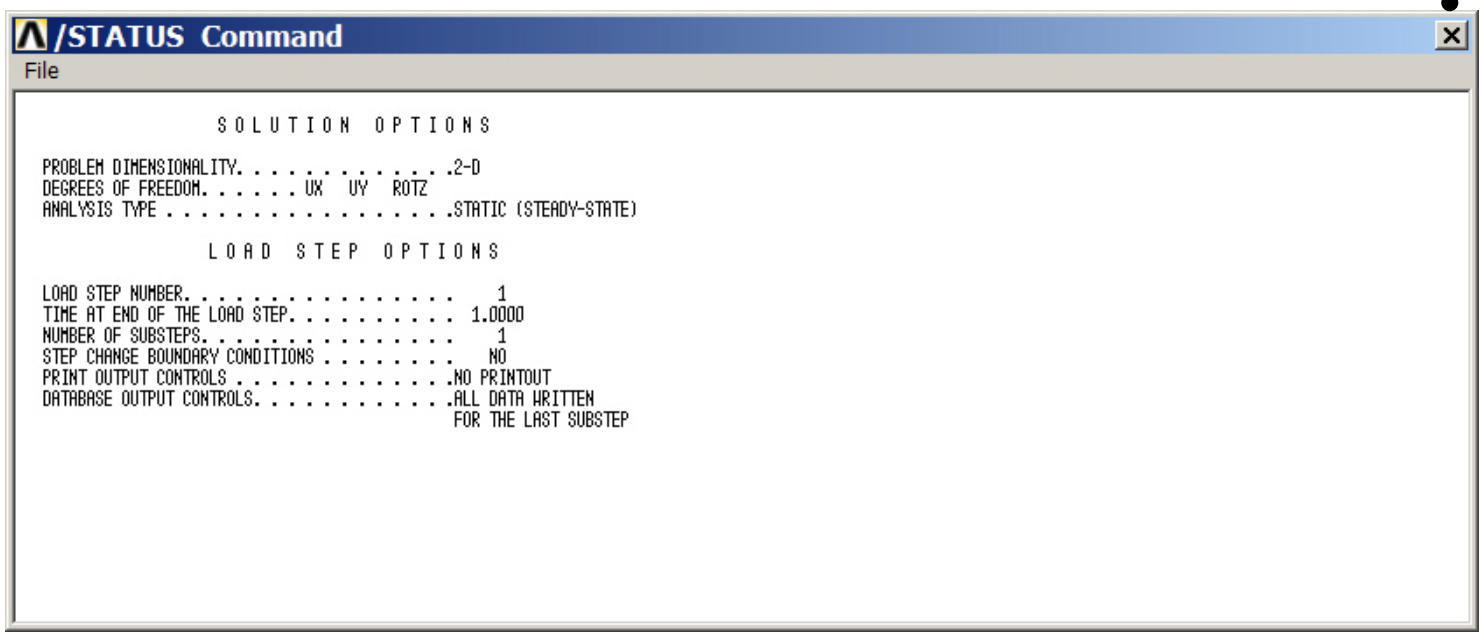
Press OK

Example - Solve



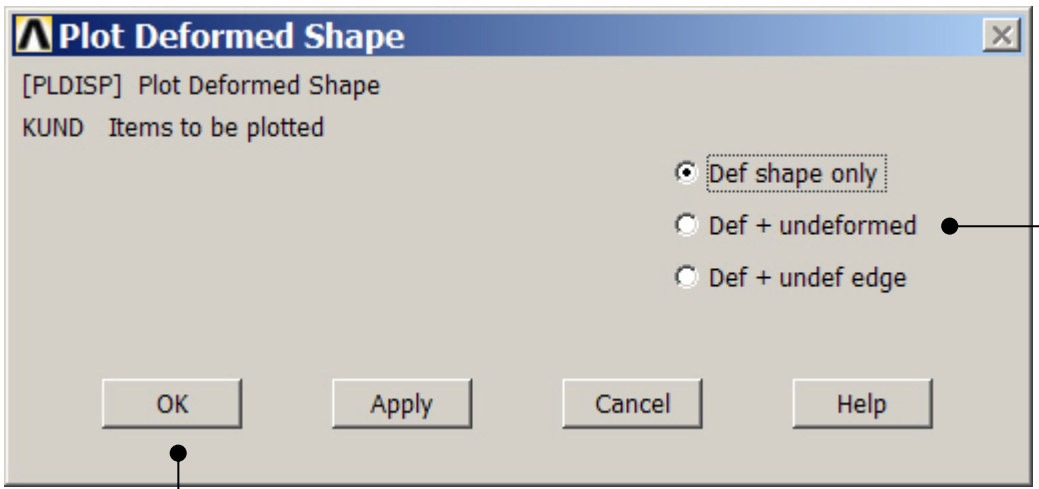
Press Close

Press here
to Close



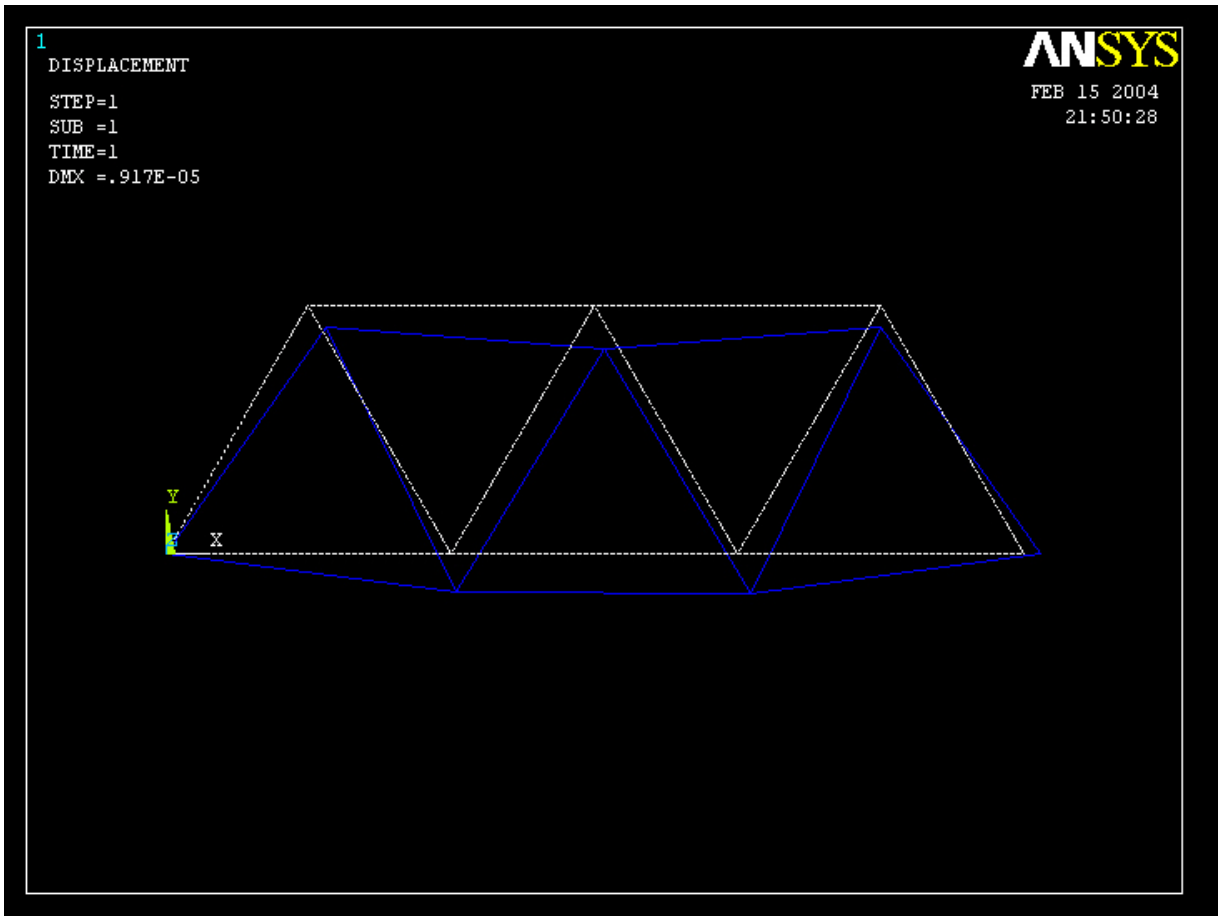
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



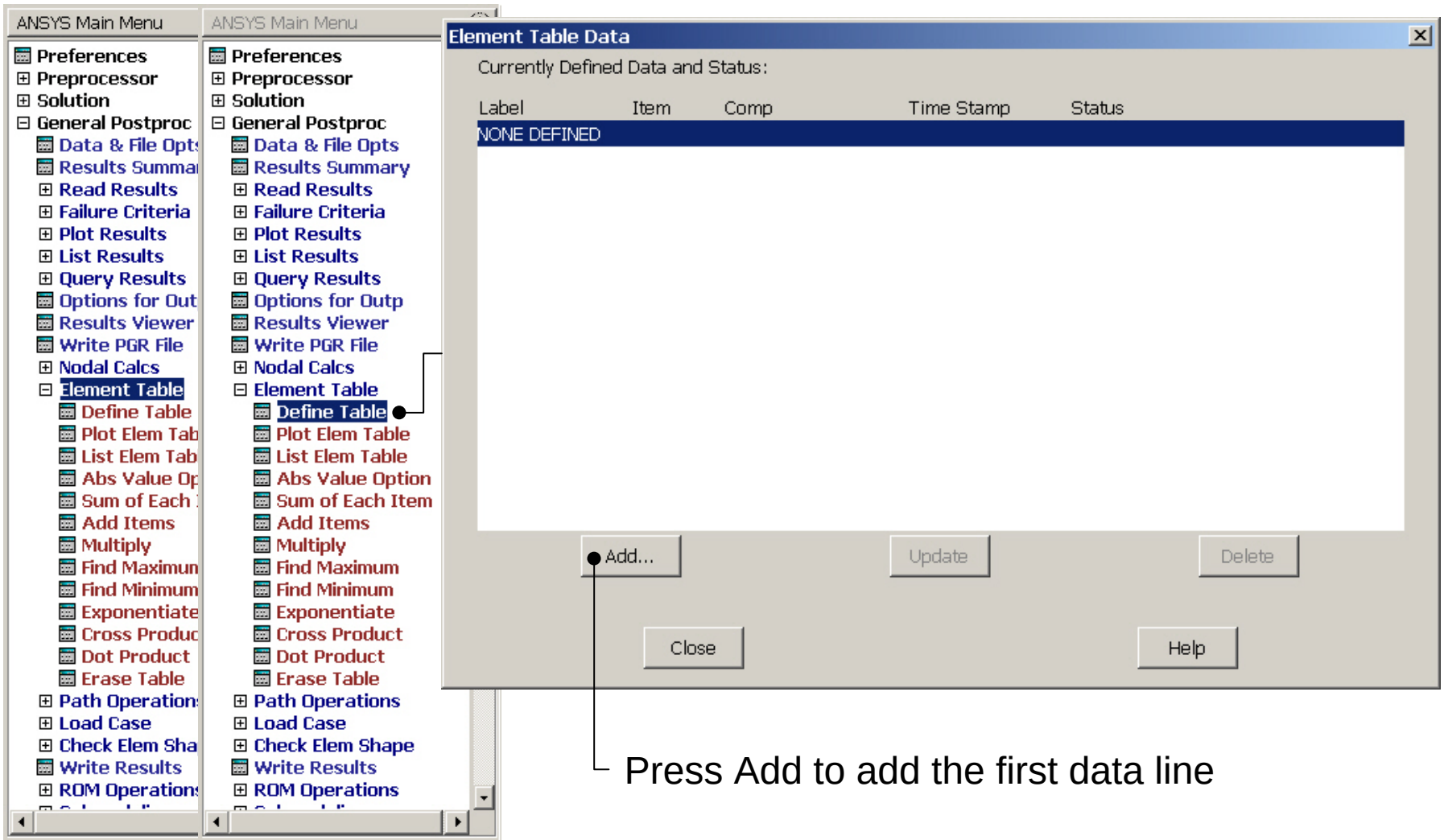
Select “Def+undeformed” and Press OK

Example - PostProcessing



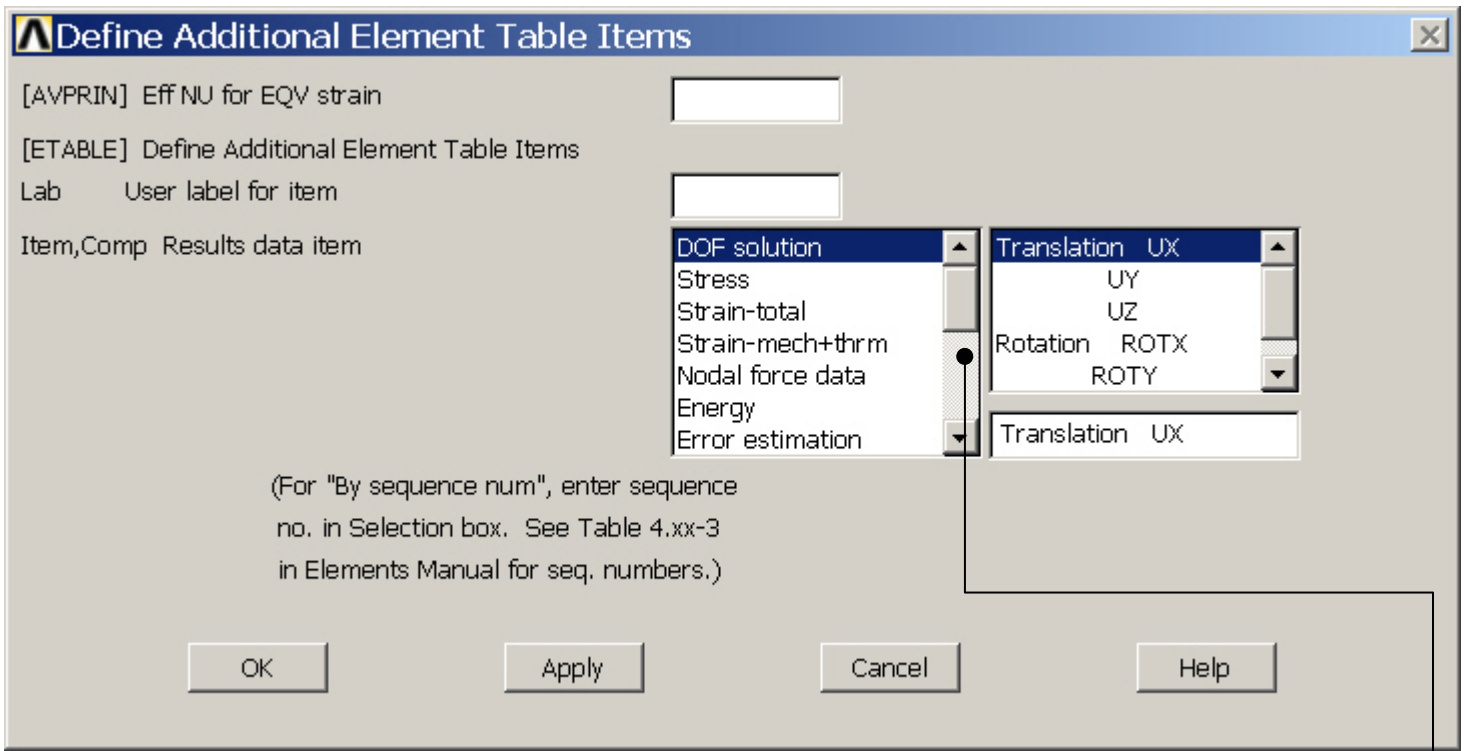
Read Maximum displacement: DMX

Example – Element Table



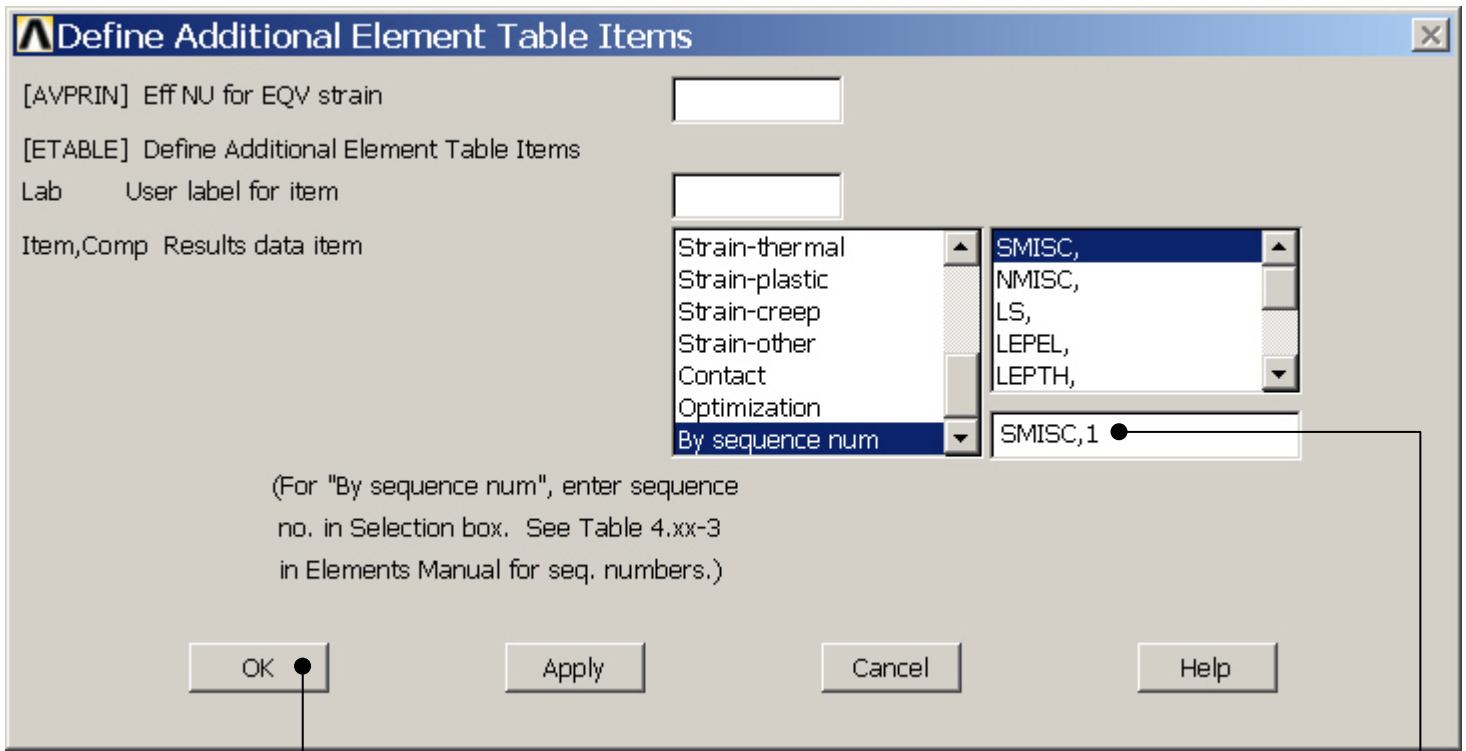
Press Add to add the first data line

Example – Element Table



Scroll down in this menu to find the line “By sequence number”

Example – Element Table

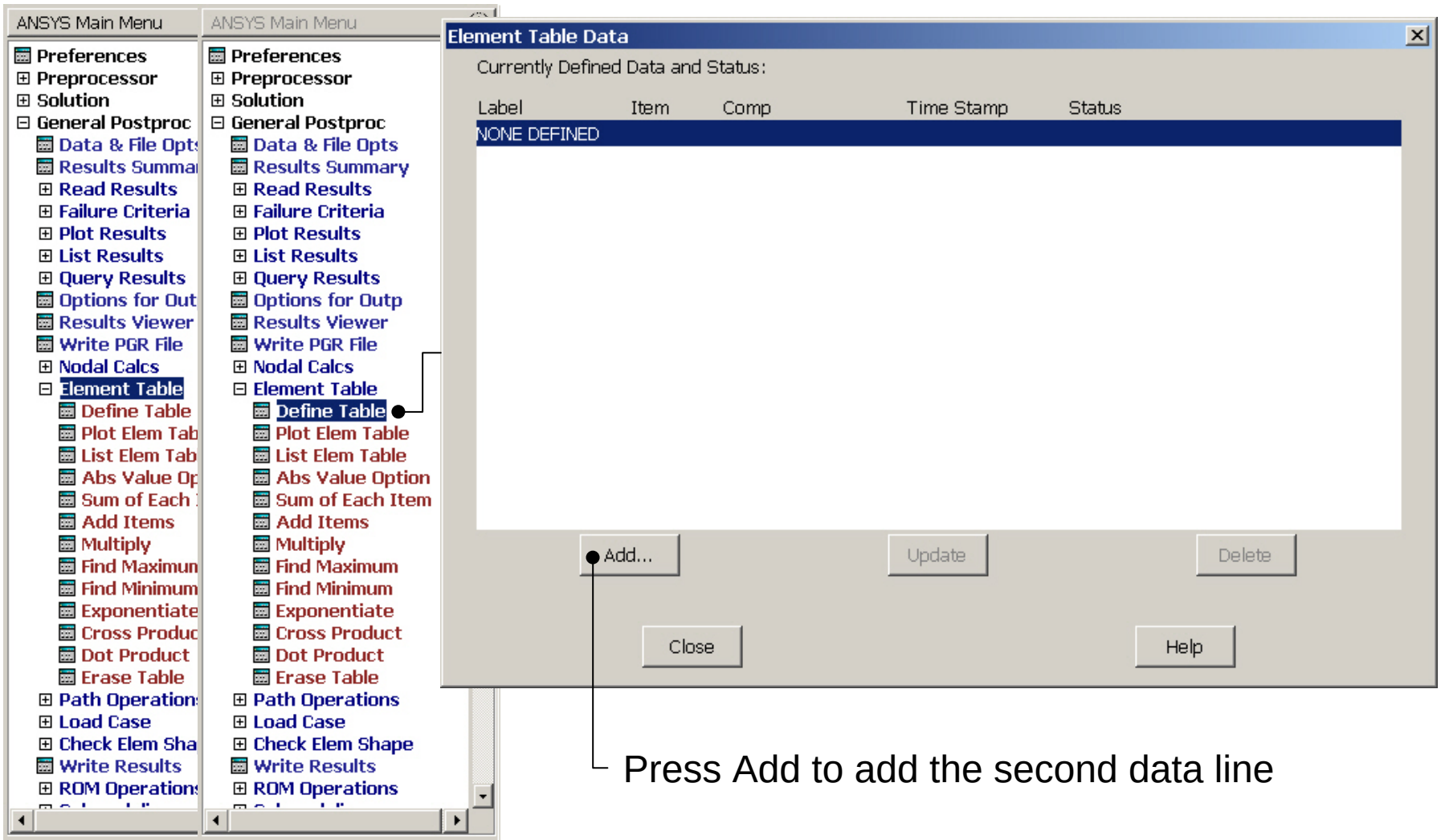


Press OK

Enter 1 as found in table 3.2

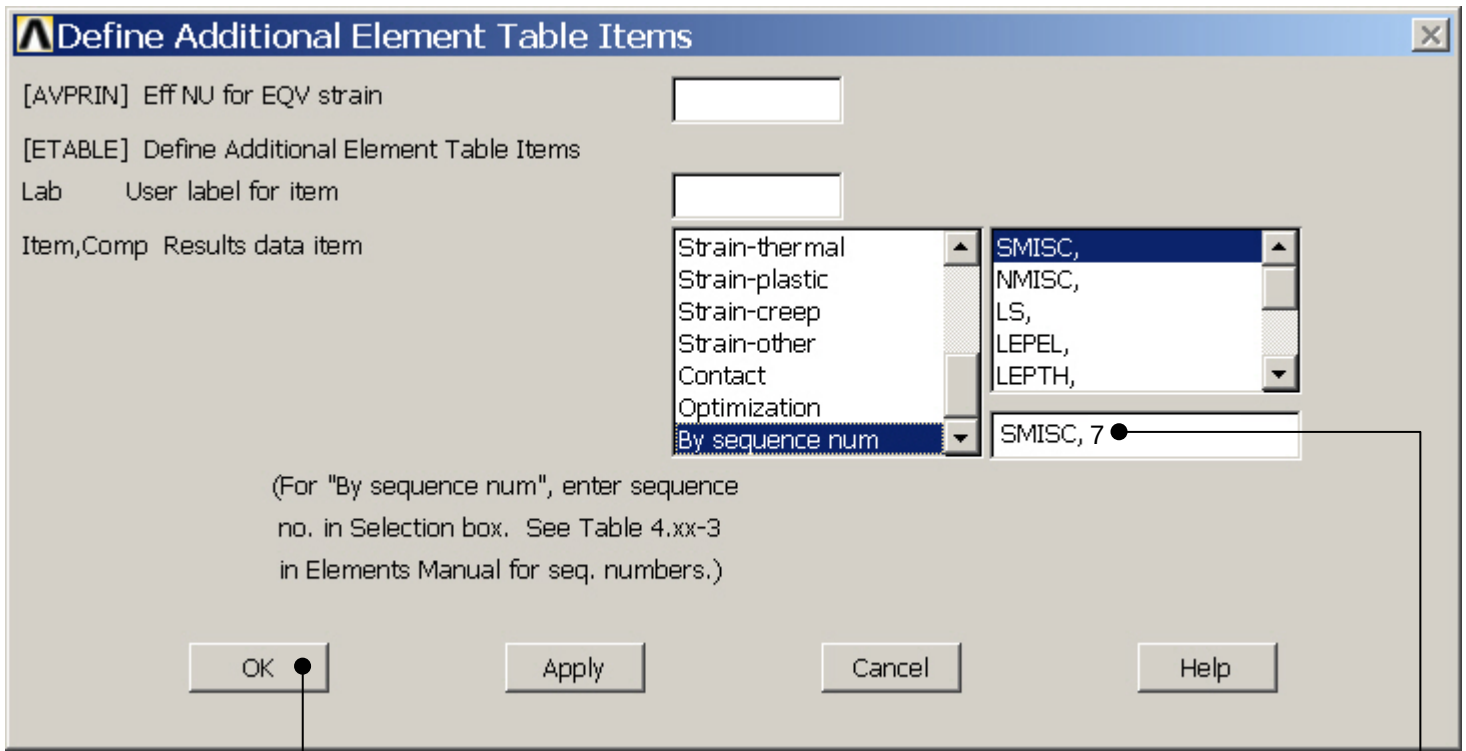
From table 3.2 MFORX, SMISC,1,7

Example – Element Table



Press Add to add the second data line

Example – Element Table

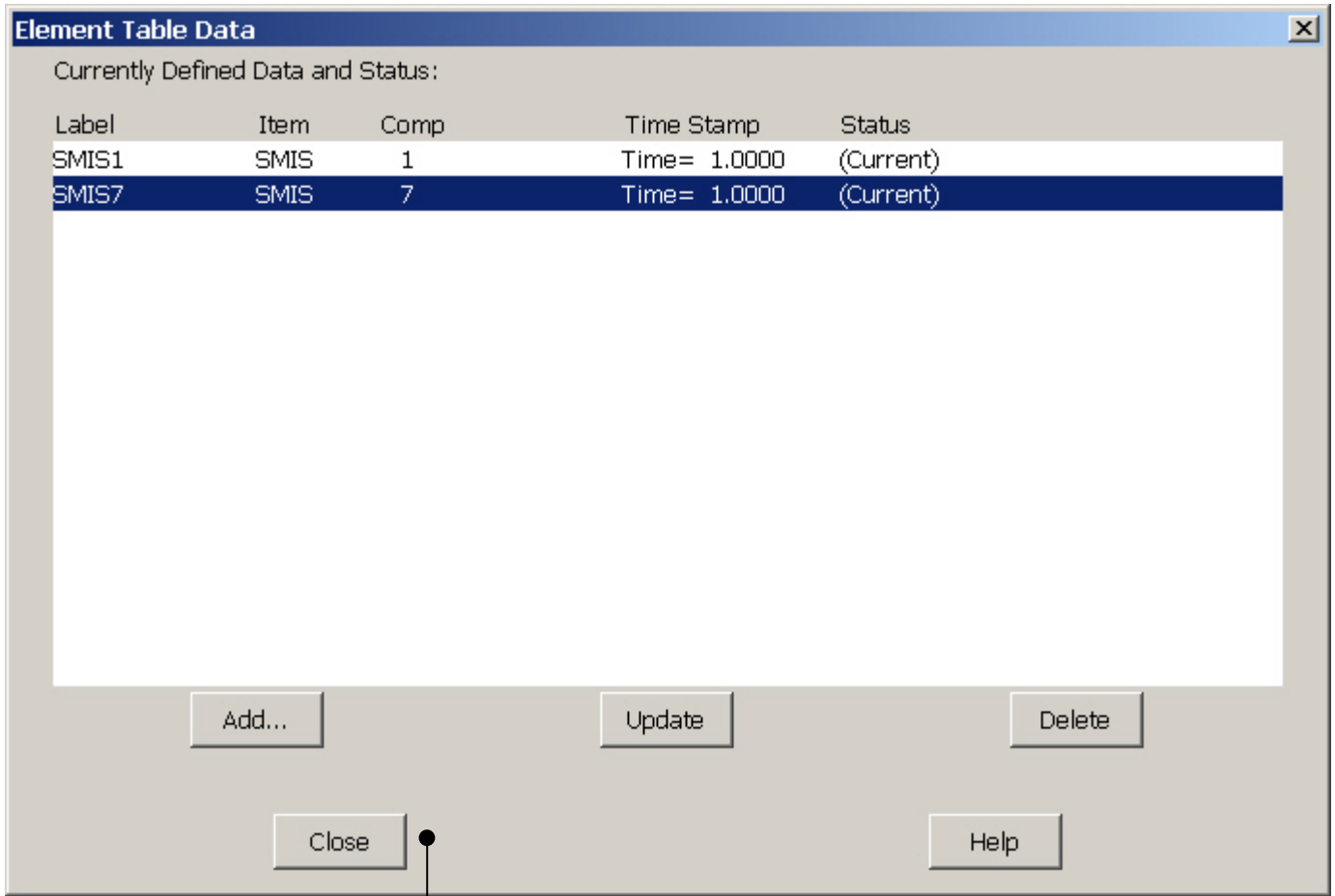


Press OK

Enter 7 as found in table 3.2

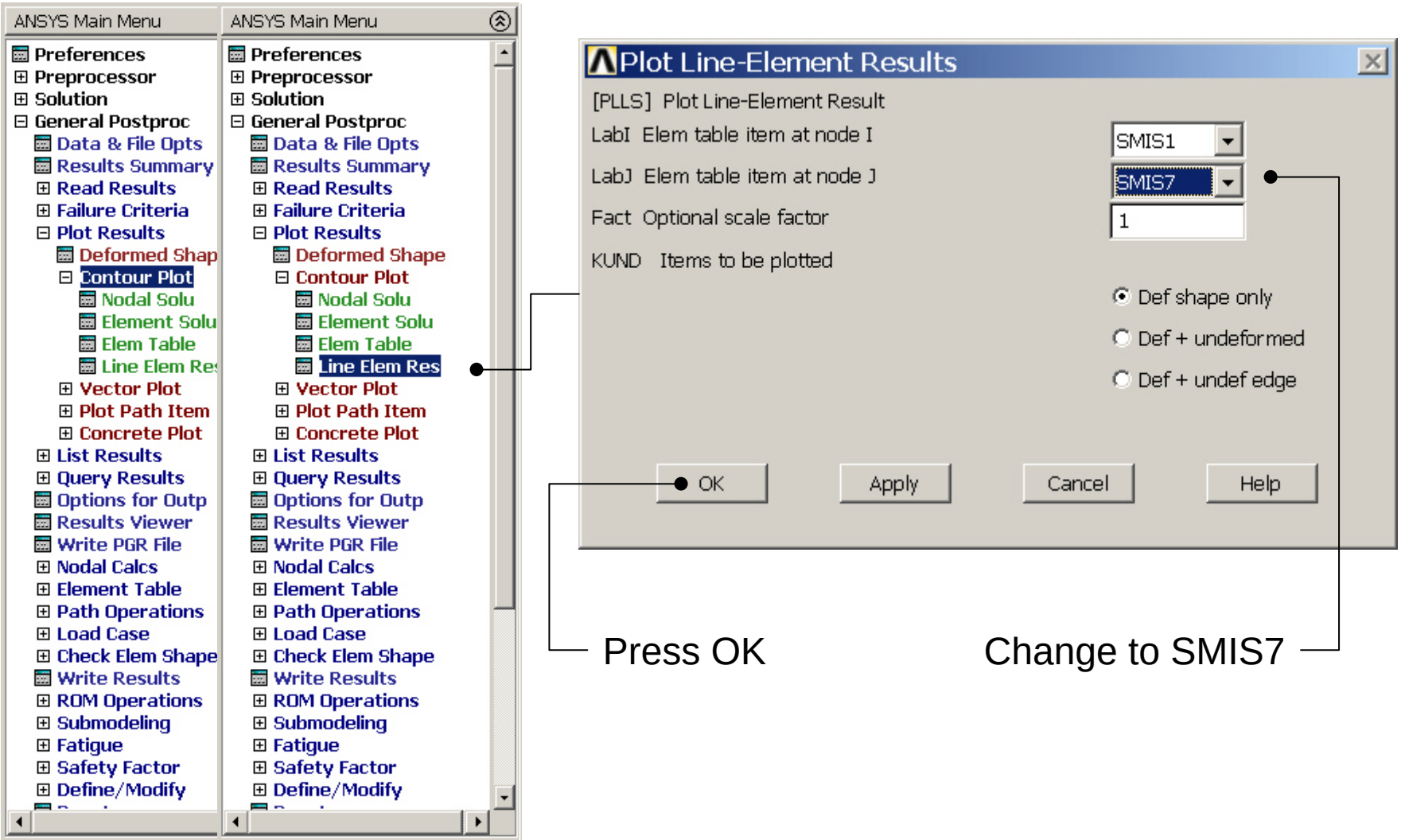
From table 3.2 MFORX, SMISC,1,7

Example – Element Table

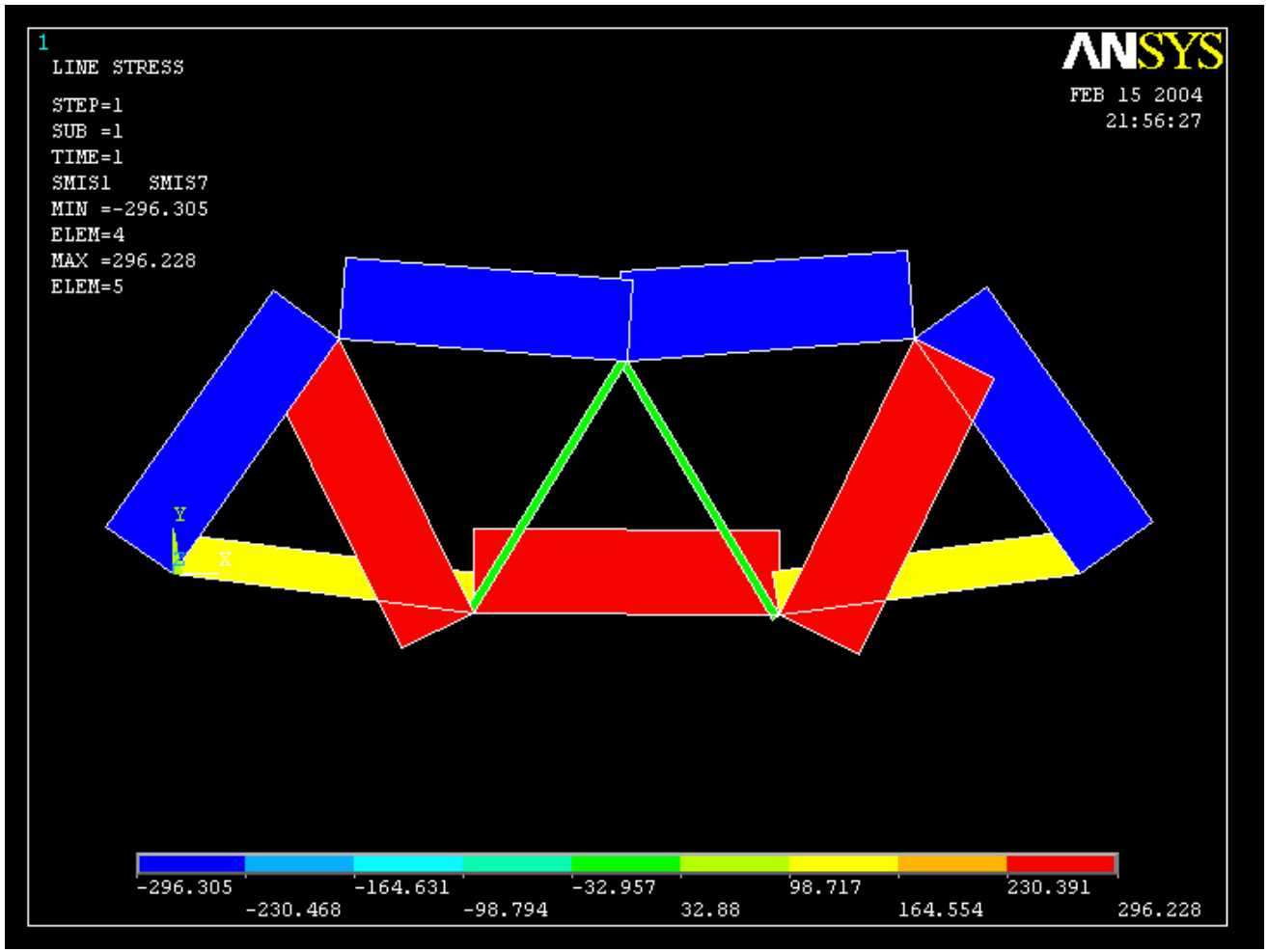


Press Close

Example – Plot Line-Element



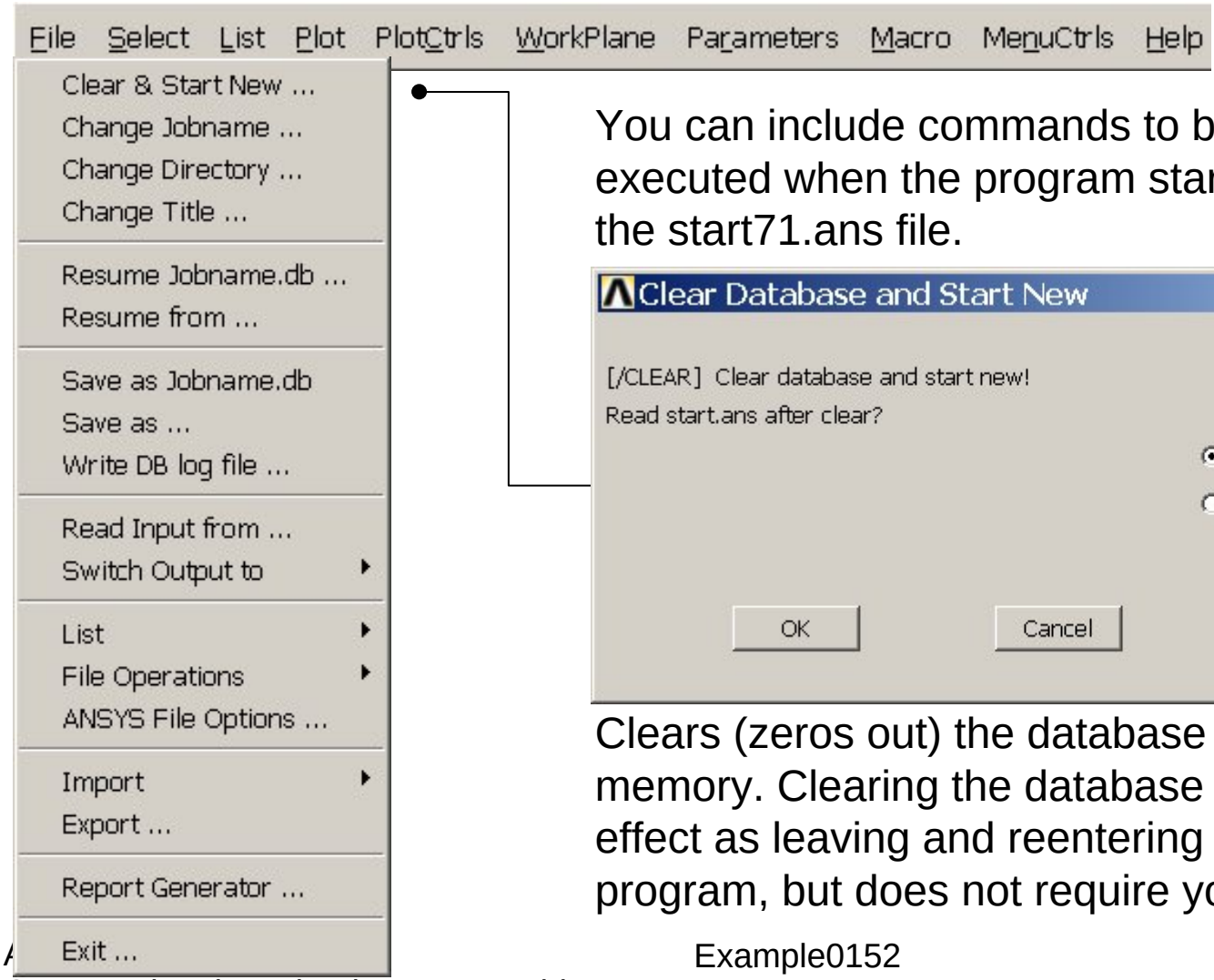
Example – Plot Line-Element



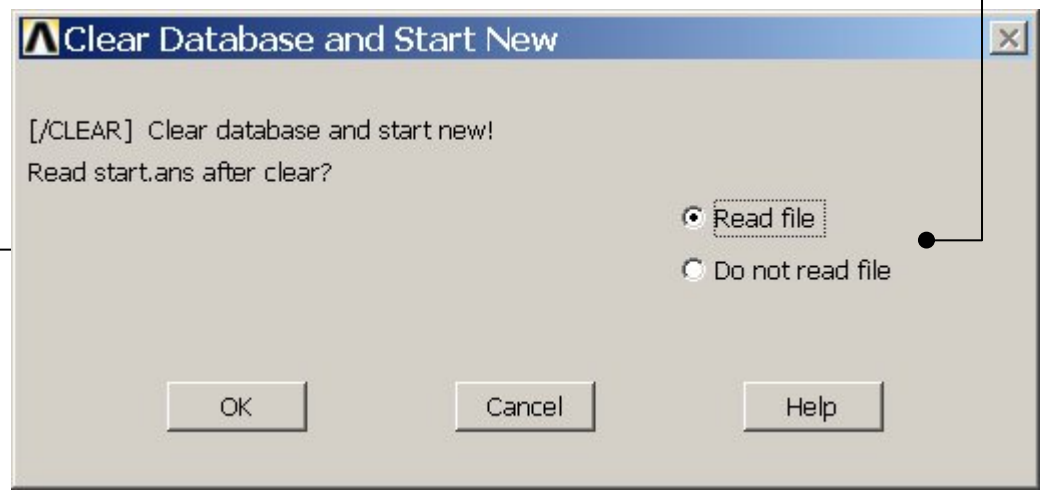
Example – Comments/Questions

- Try Link elements instead of beam elements?
- The “example0152.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.