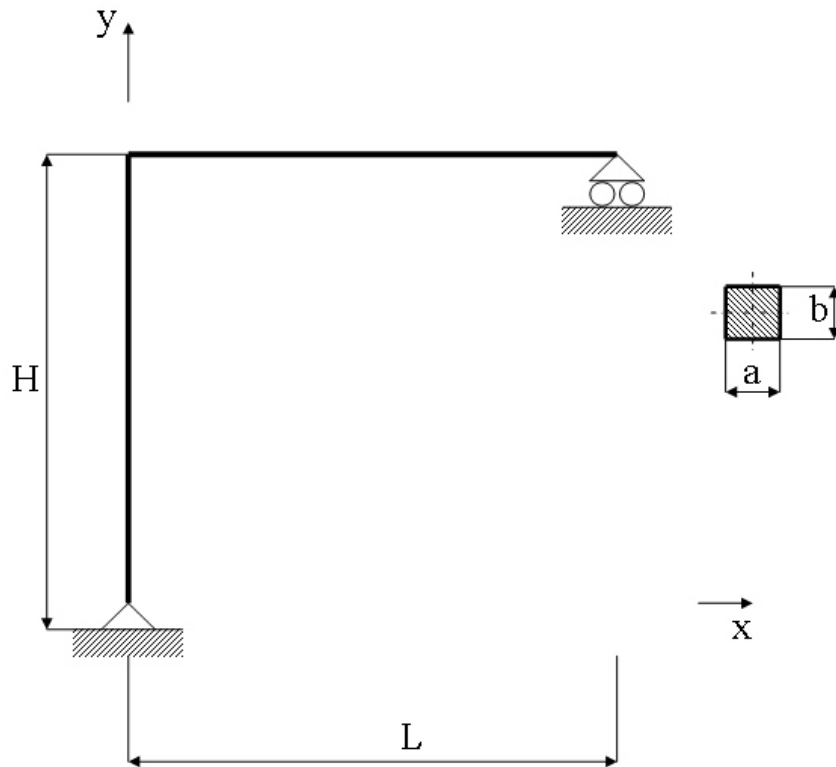


Course in ANSYS

Example0410

Example – Frame 2D



Objective:

Compute the harmonic response

Tasks:

Perform a modal analysis

Display the mode shapes

Perform a harmonic analysis

Topics:

Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, modal and harmonic analysis

$$E = 200 \text{E}9 \text{N/m}^2$$

$$\nu = 0.3$$

$$L = 2 \text{m}$$

$$H = 3 \text{m}$$

$$I = 0.1^4/12 \text{m}^4$$

$$\rho = 7860 \text{kg/m}^3$$

Example - title

Utility Menu > File > Change Jobname

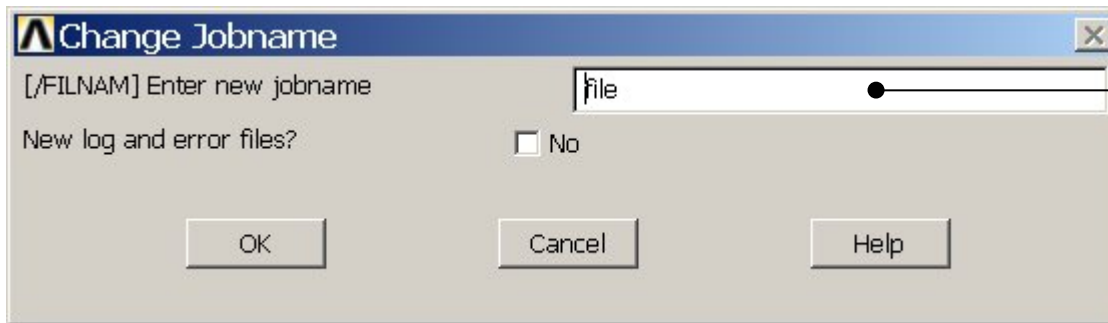


GUI

/jobname, Example0410



Command line entry

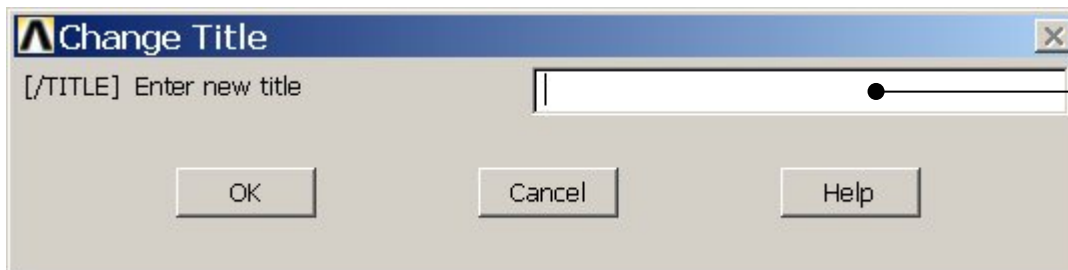


Enter: Example0410

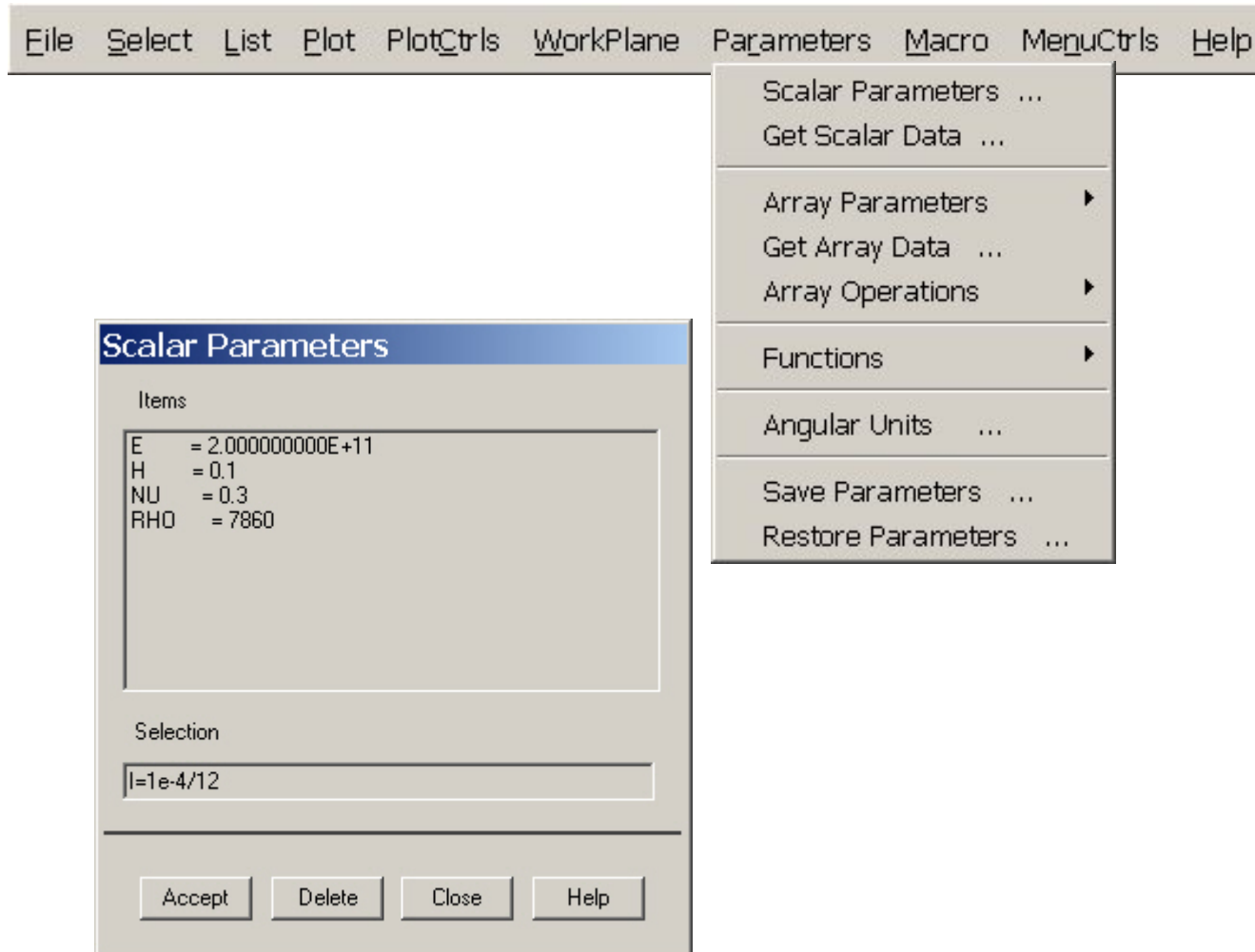
Utility Menu > File > Change Title

/title, Frame 2D

Enter: Frame 2D



Example - Parameters



Example - Keypoints

Note: An empty #
result in automatic
numbering.

Preprocessor > Modeling > Create > Keypoints > In Active CS

/PREP7

K,,,,

K,,,3,

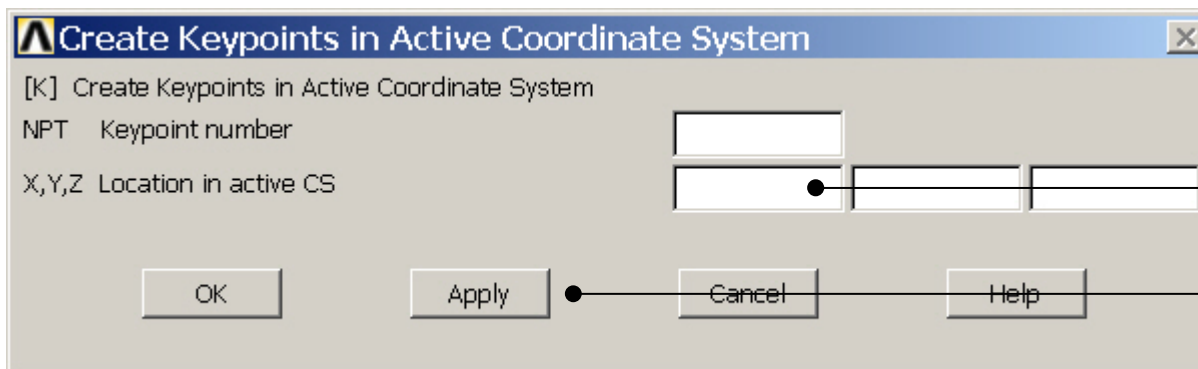
K,,2,3,

General format:

K,#,X,Y,Z

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

Enter 0,0,0 and
Press **Apply**
Enter 0,3,0 and
Press **Apply**
Enter 2,3,0 and
Press **Apply**

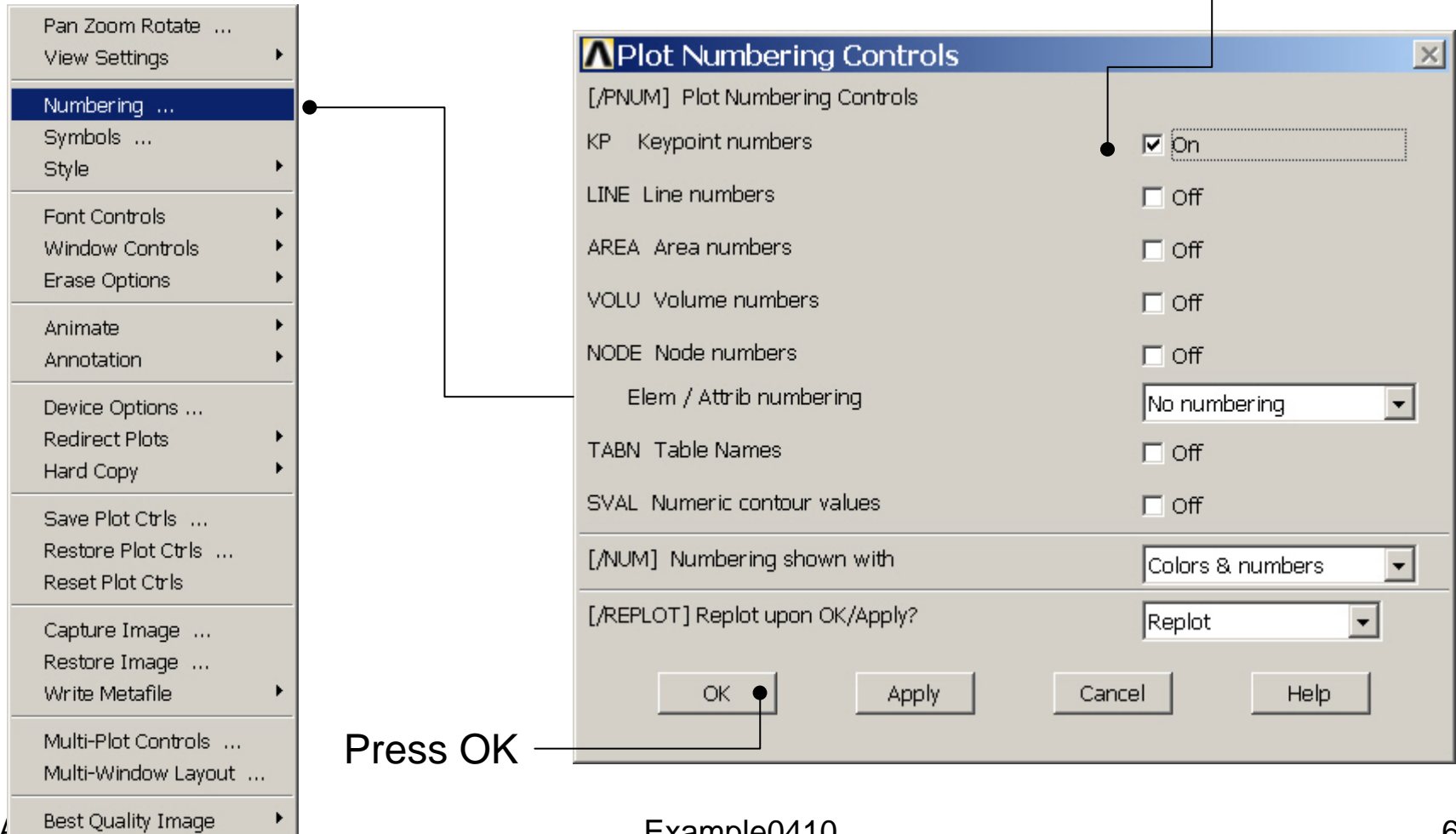


Note: An empty box
result in a zero. It is
allowed to enter 0.0
in each box.

Example - Numbering

Utility Menu > PlotCtrls > Numbering

Switch on Keypoint numbers



Example0410

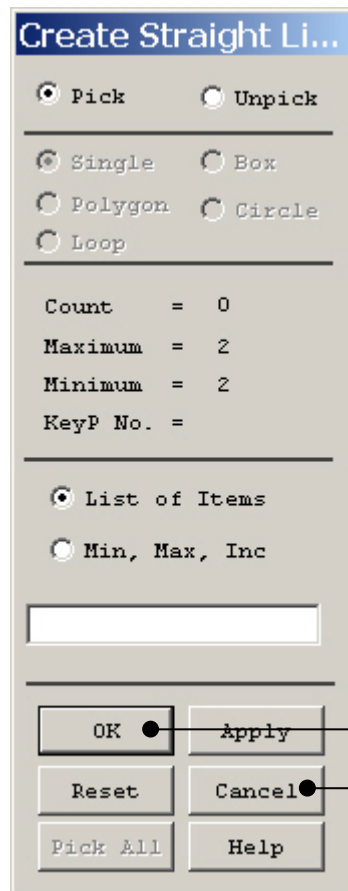
Example - Lines

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

Create a line between KP1 and KP2 and KP2 and KP3.

L,1,2

L,2,3



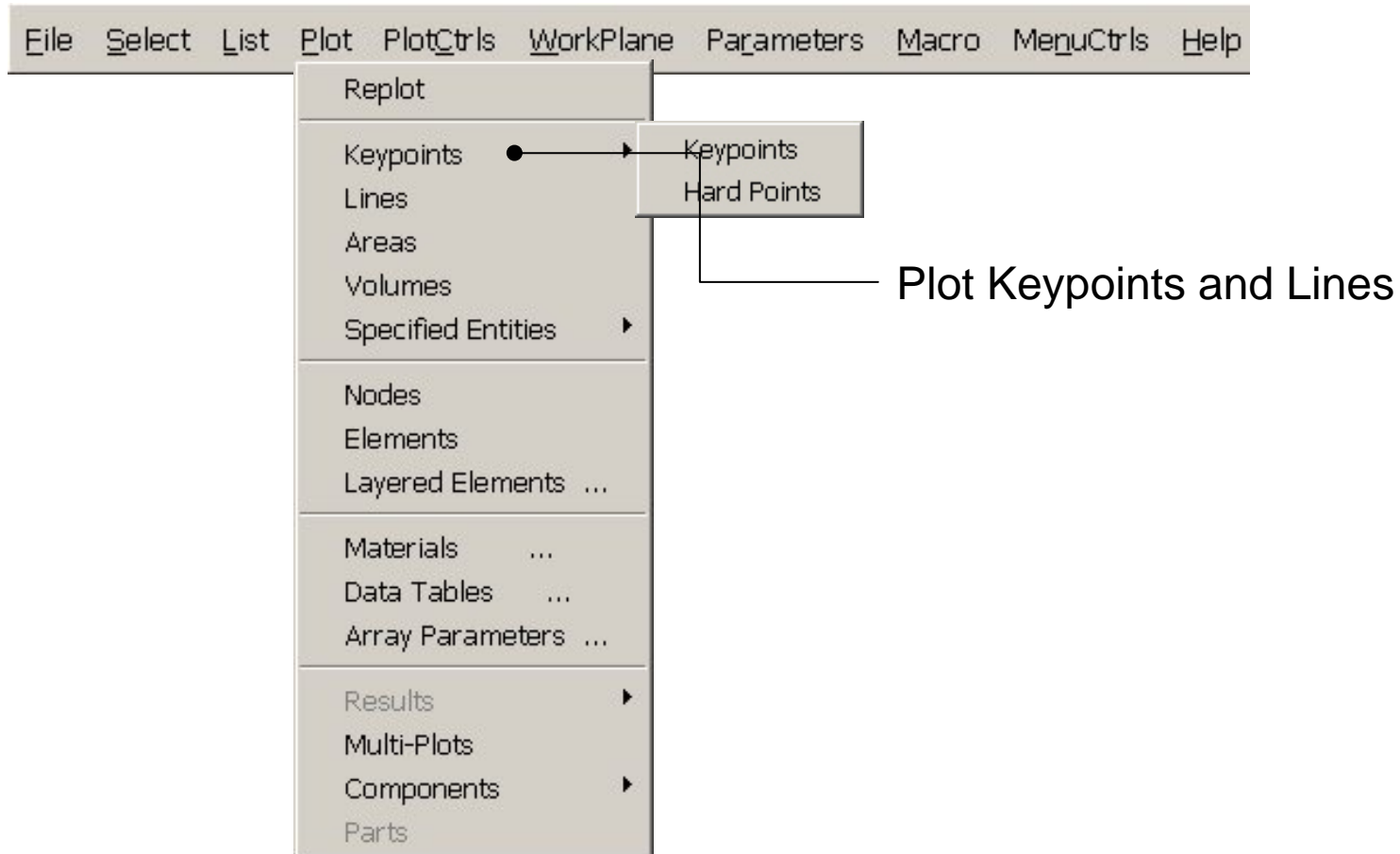
HINT: By clicking with the right-hand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

Pick: upward arrow

Unpick: downward arrow

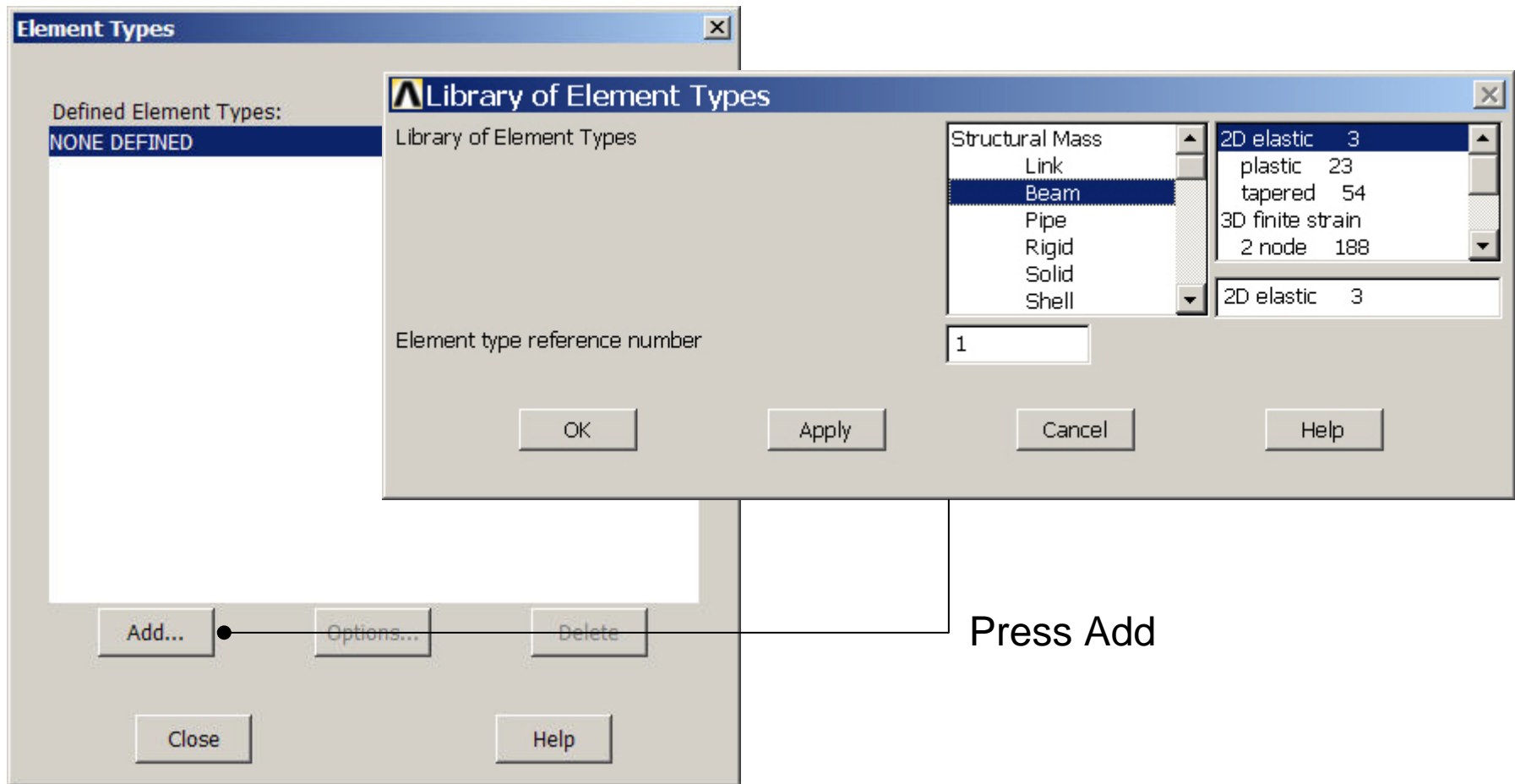
Press OK or Cancel to finish selection

Example - Plot - Nodes



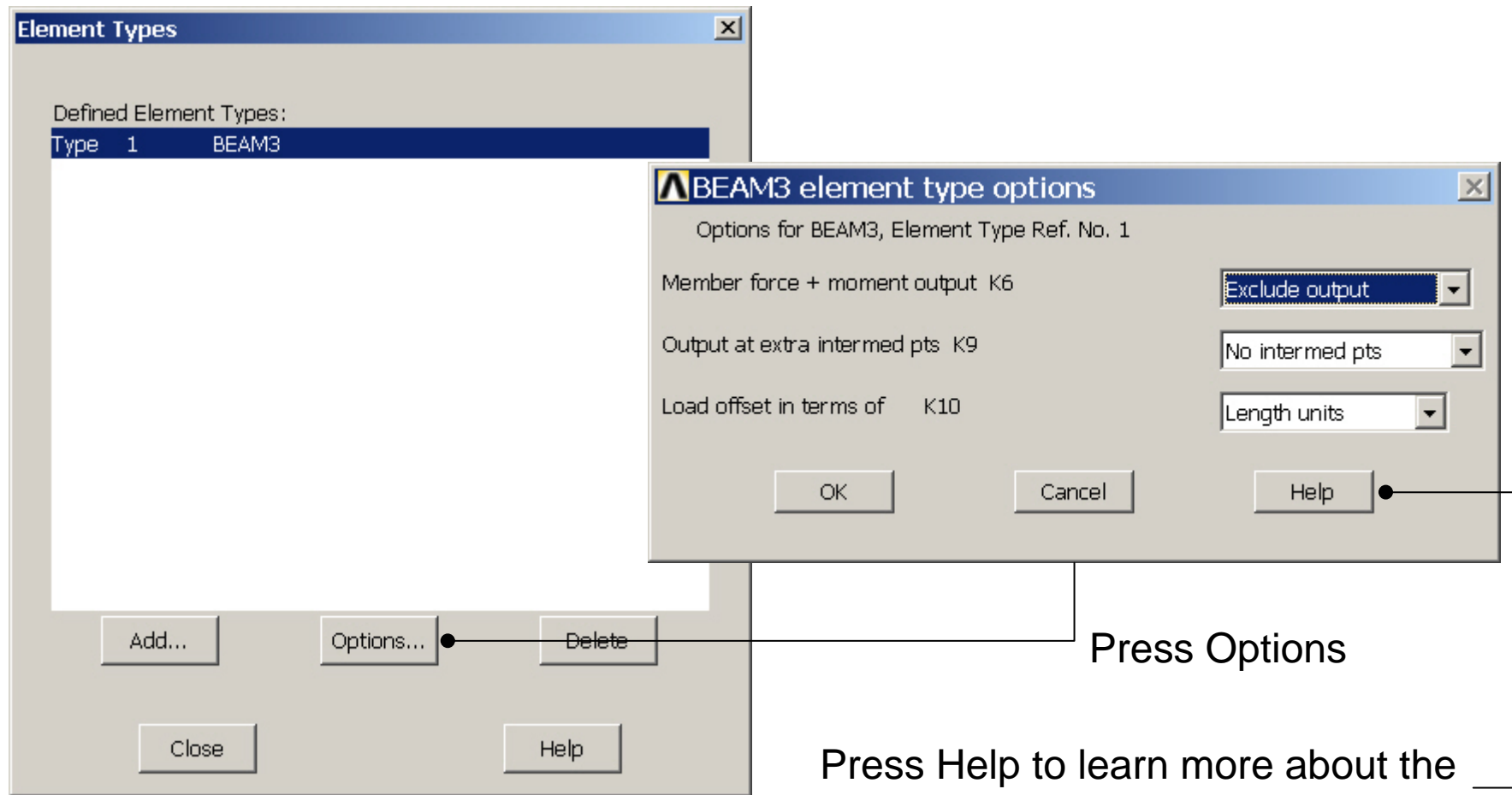
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



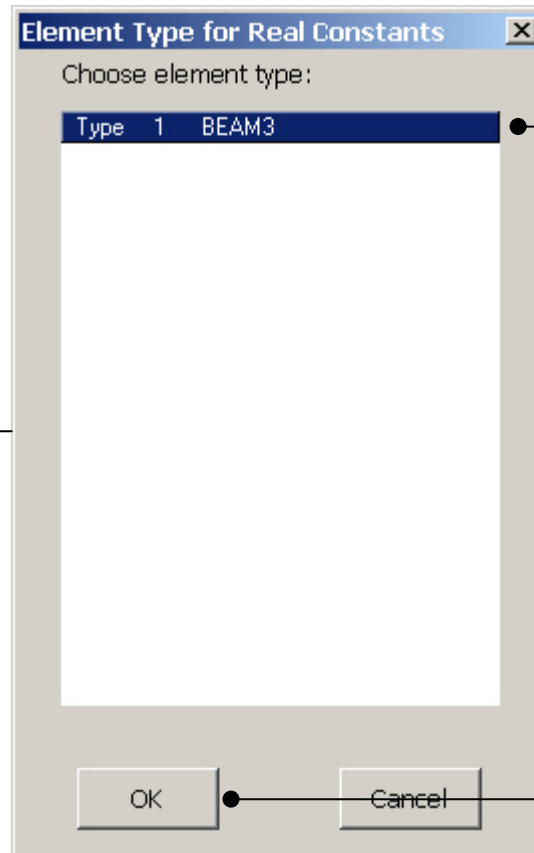
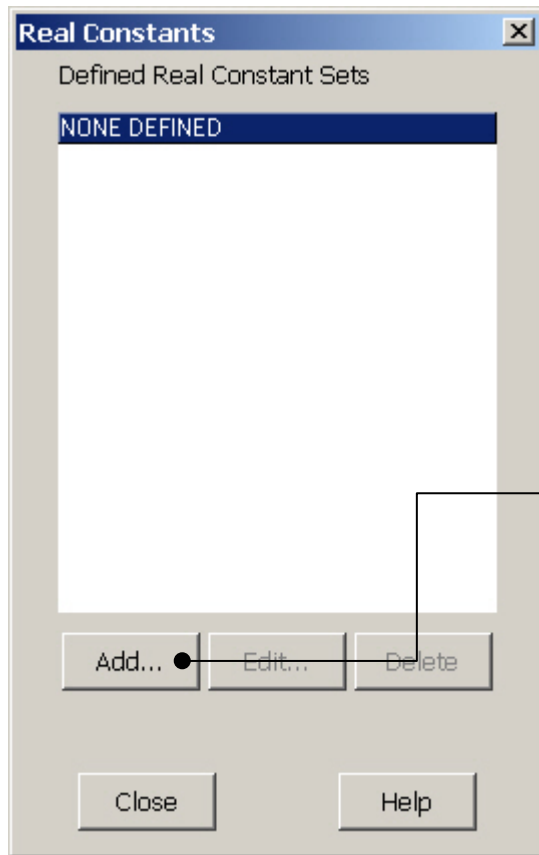
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Real Constants

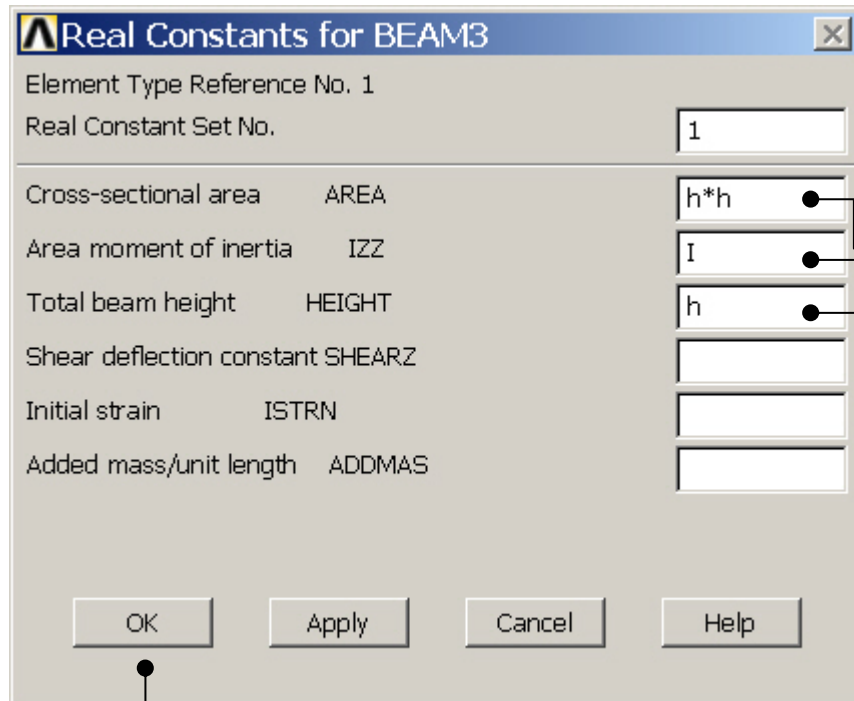
Preprocessor > Real Constants > Add



Place the cursor on the relevant element and press OK

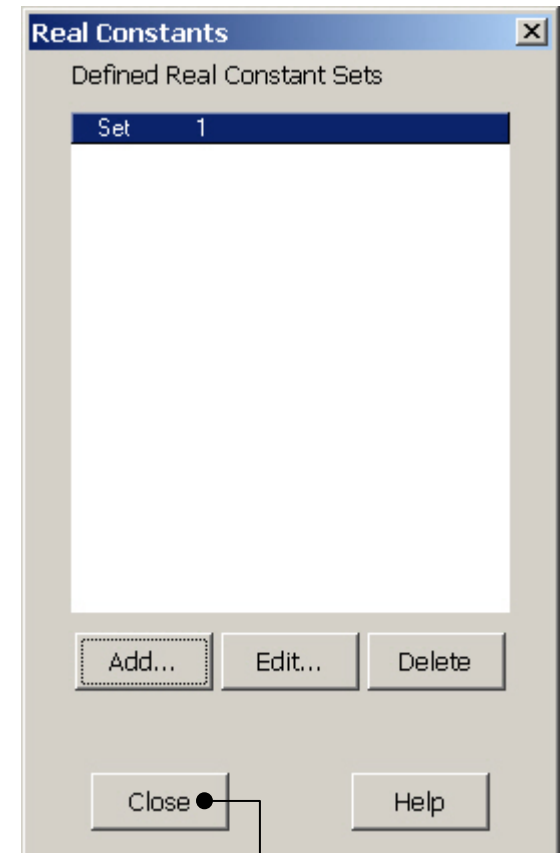
Example - Real Constants

Preprocessor > Real Constants > Add



Enter cross sectional data

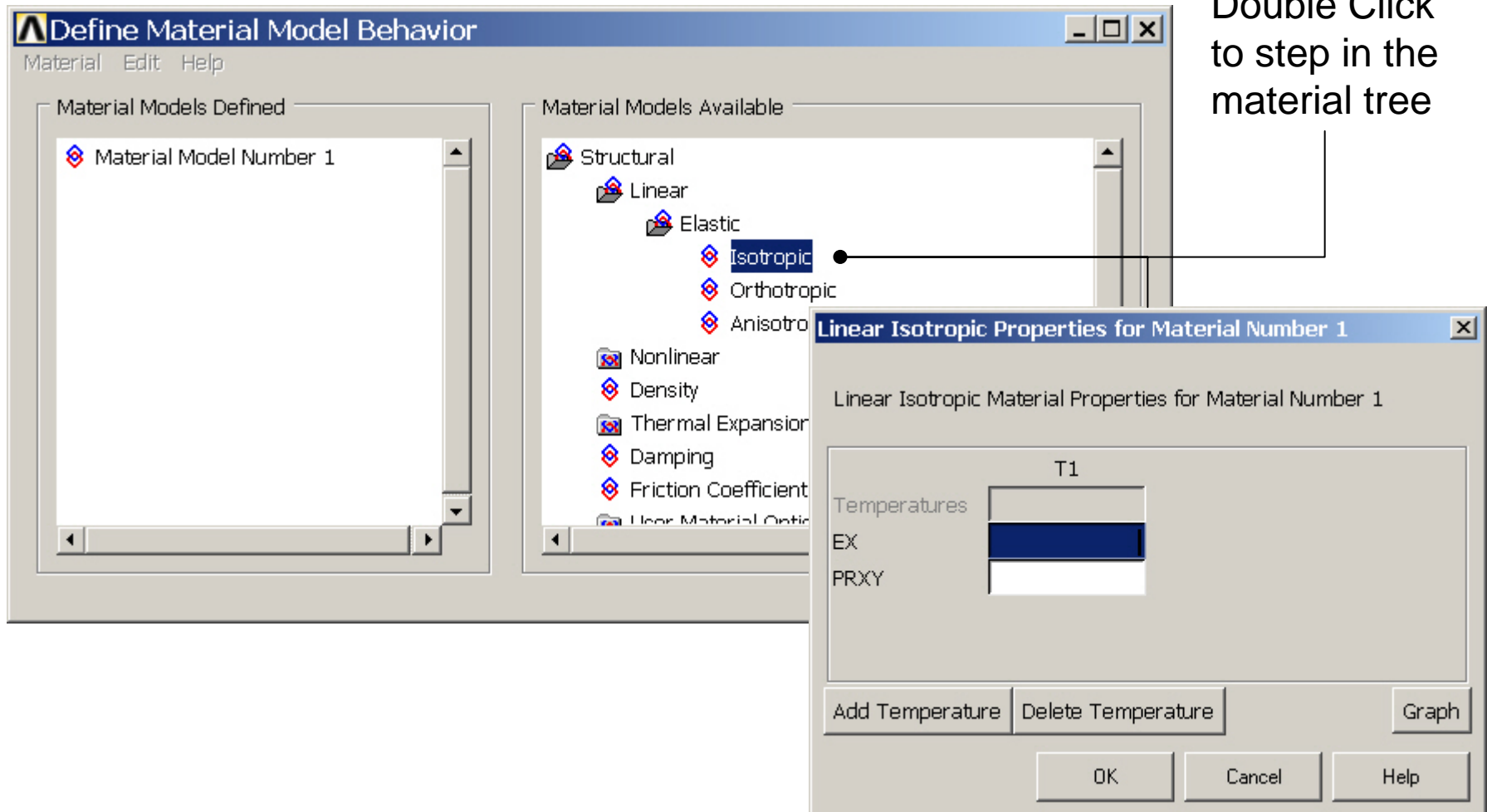
Press OK



Press Close to finish

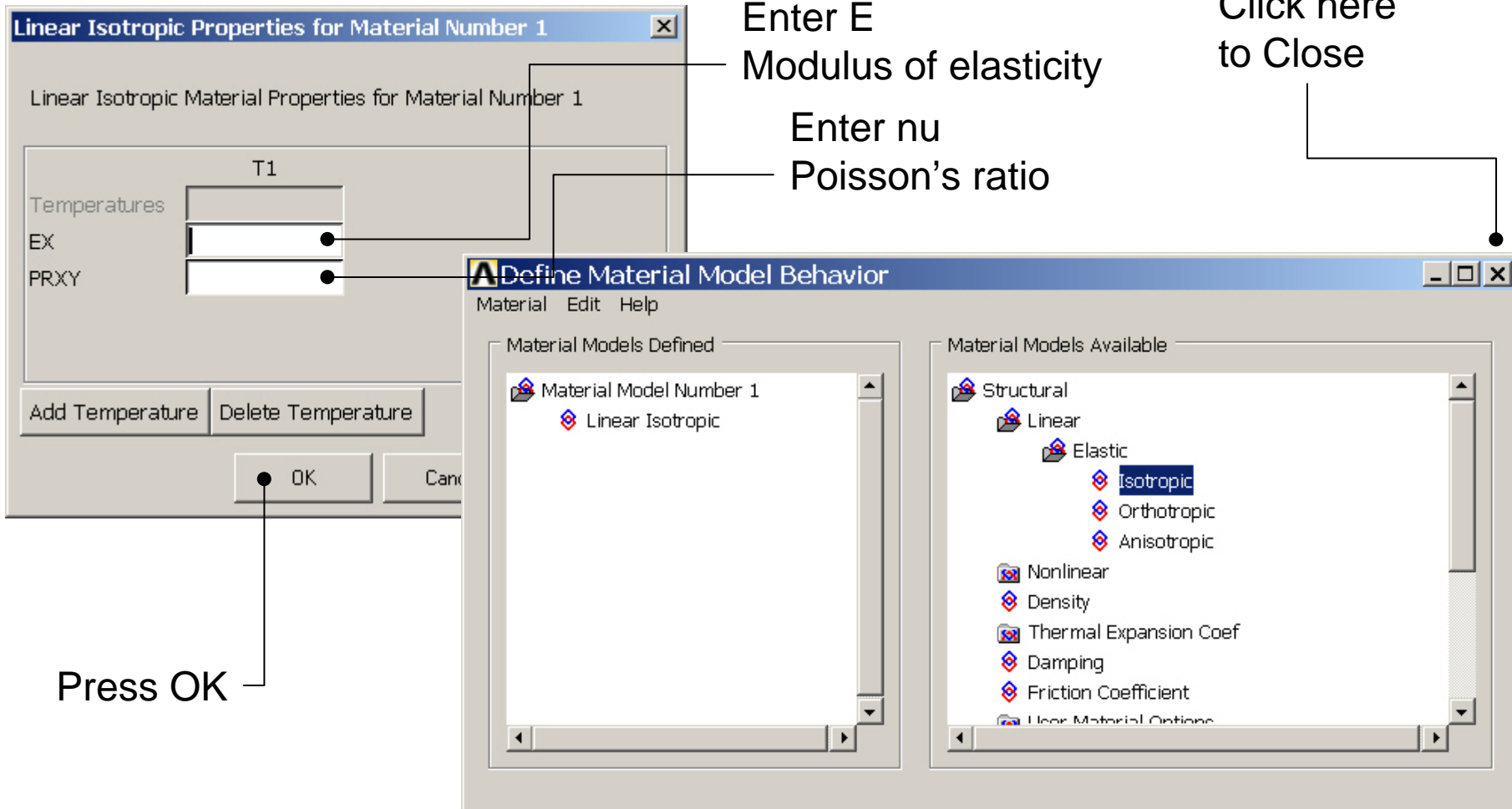
Example - Material Properties

Preprocessor > Material Props > Material Models

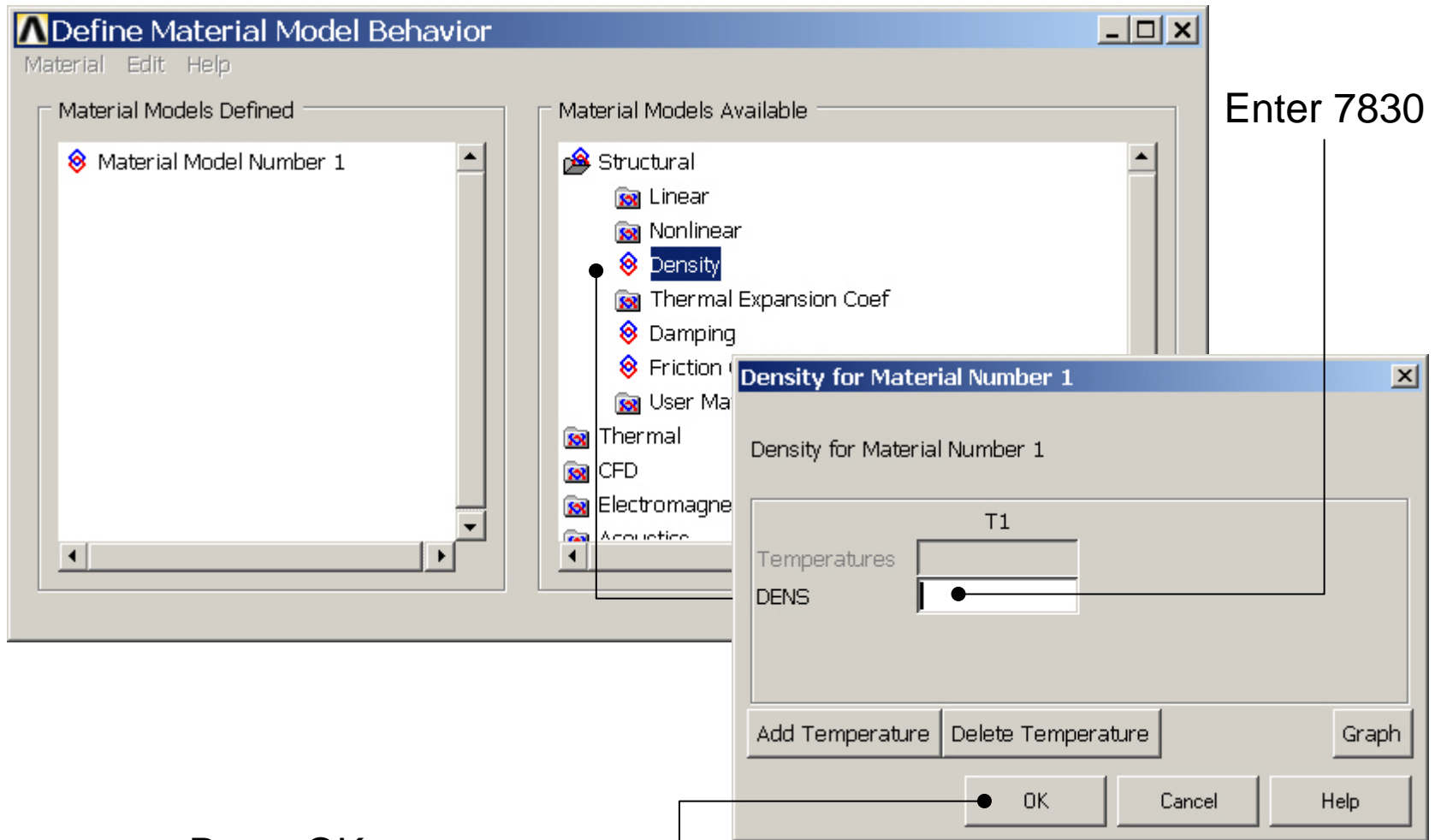


Example - Material Properties

Preprocessor > Material Props > Material Models



Example - Density



Example - Meshing

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

Select/Pick
L1

Element Size on Picked Lines

☒ 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 Help

Press OK when finish with selection

Enter 30

Example - Meshing

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

Select/Pick
L2

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

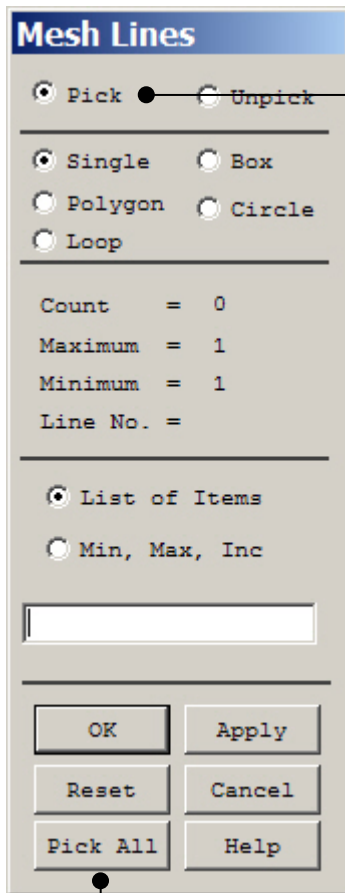
OK Apply Cancel Help

Press OK when finish with selection

Enter 20

Example - Meshing

Preprocessor > Meshing > Mesh > Lines



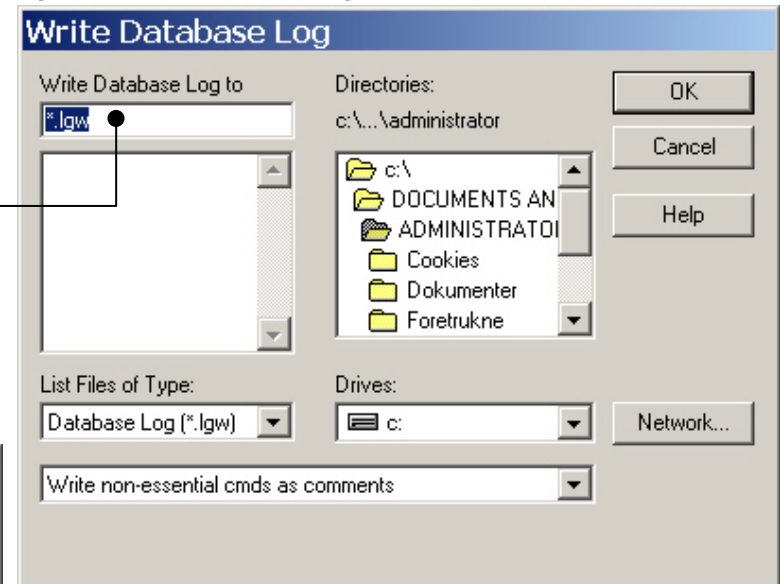
Select individual lines to be meshed by Picking

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

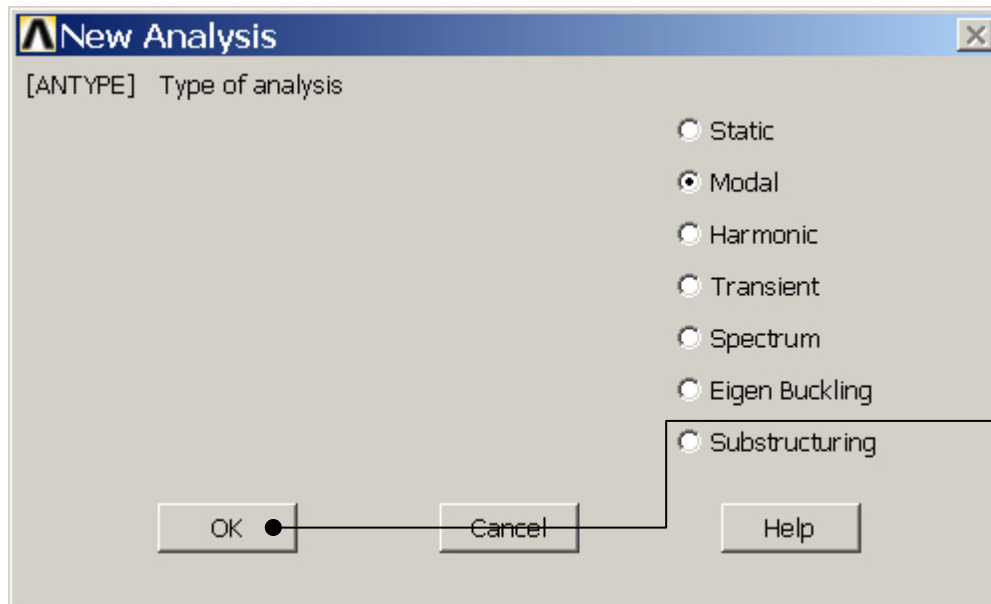
Select all lines defined to be meshed

Example – Analysis Type

File > Write DB log file
Enter “example0410.lgw”



Solution > Analysis Type > New Analysis



Press OK

Example – MA Analysis Options

The image shows the ANSYS Main Menu on the left and the Modal Analysis dialog box on the right. The dialog box is titled "Modal Analysis" and contains several sections for configuring the analysis options. Annotations with lines pointing to specific fields provide instructions for each step.

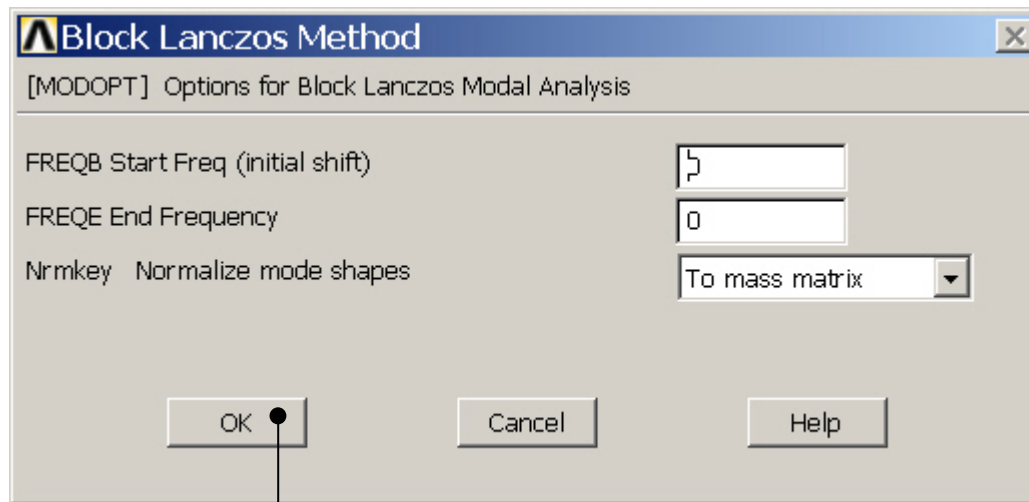
ANSYS Main Menu:

- Preferences
- Preprocessor
- Solution
 - Analysis Type
 - New Analysis
 - ExpansionPass
 - Analysis Options
 - Define Loads
 - Load Step Opts
 - Solve
 - FSI Set Up
 - Unabridged Menu
- General Postproc
- TimeHist Postpro
- Topological Opt
- RDM Tool
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

Modal Analysis Dialog Box:

- [MODEPT] Mode extraction method:**
 - ☒ Block Lanczos (Select Block Lanczos)
 - ☐ Subspace
 - ☐ Powerdynamics
 - ☐ Reduced
 - ☐ Unsymmetric
 - ☐ Damped
 - ☐ QR Damped
- No. of modes to extract:** 0 (Enter 10)
(must be specified for all methods except the Reduced method)
- [MXPAND] Expand mode shapes:** ☒ Yes (Switch to Yes)
 - NMODE No. of modes to expand:** 0 (Enter 10)
 - Elcalc Calculate elem results?:** ☐ No
- [LUMPM] Use lumped mass approx?:** ☐ No
-For Powerdynamics lumped mass approx will be used
- [PSTRES] Incl prestress effects?:** ☐ No
- [MSAVE] Memory save:** ☐ No
-only applies if the PowerDynamics method is selected
- Buttons:** OK, Cancel, Help (Press OK)

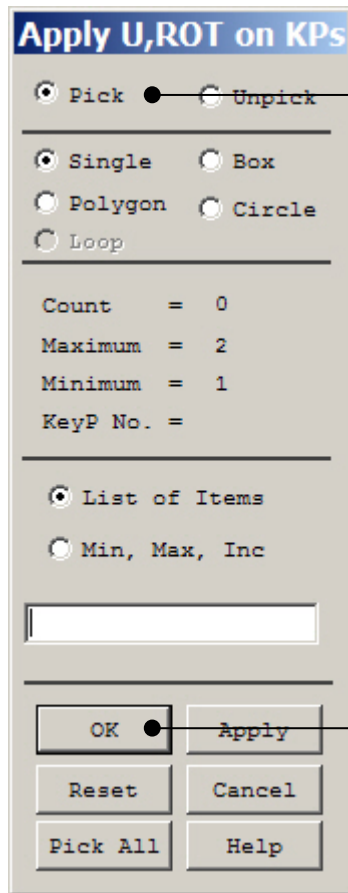
Example – MA Analysis Options



Press OK

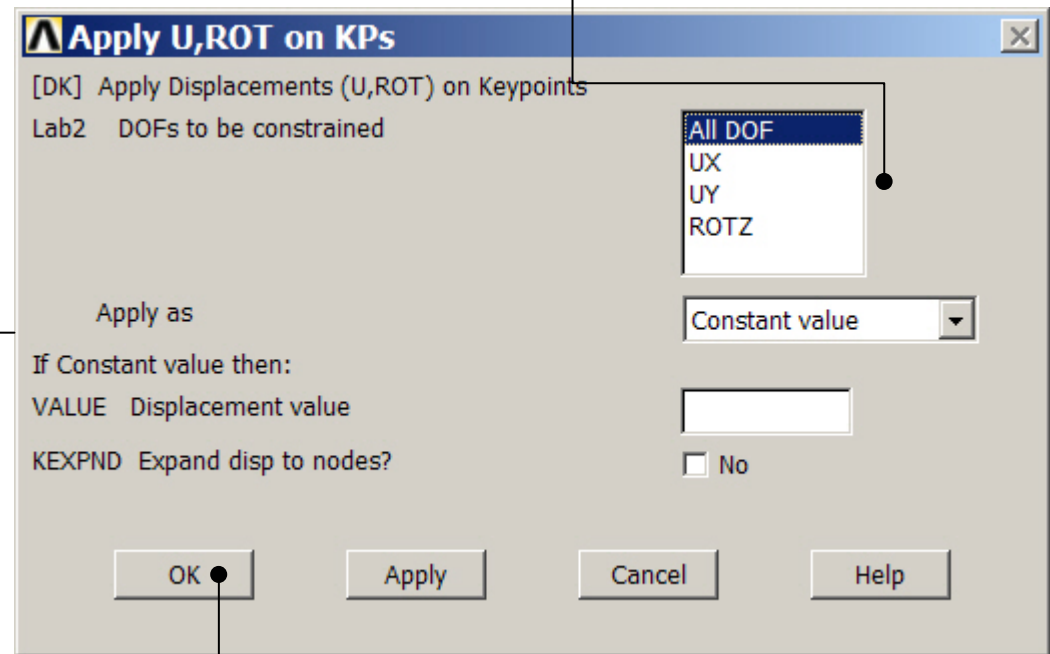
Example – Define Loads

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



Select keypoint 1

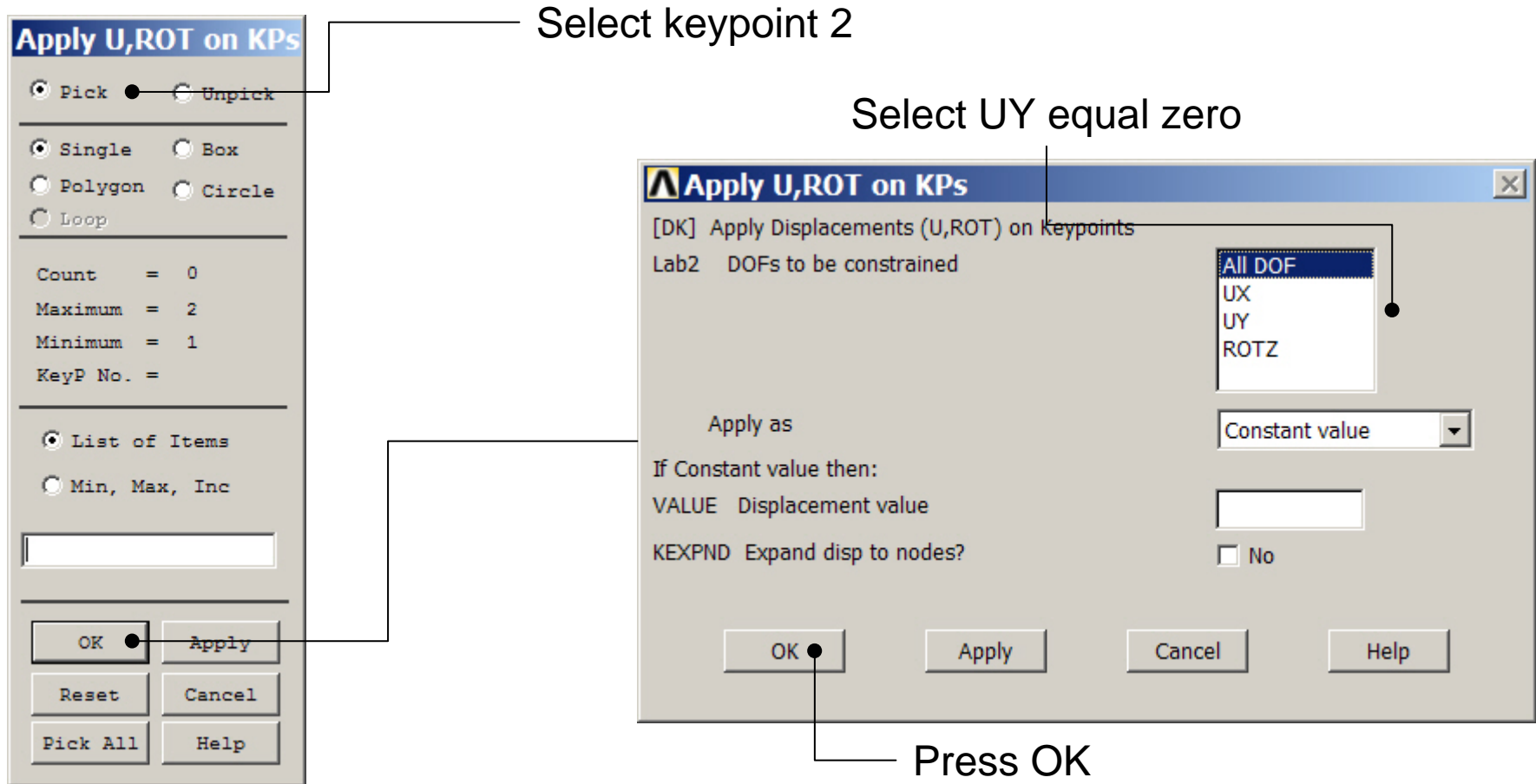
Select UX and UY equal zero



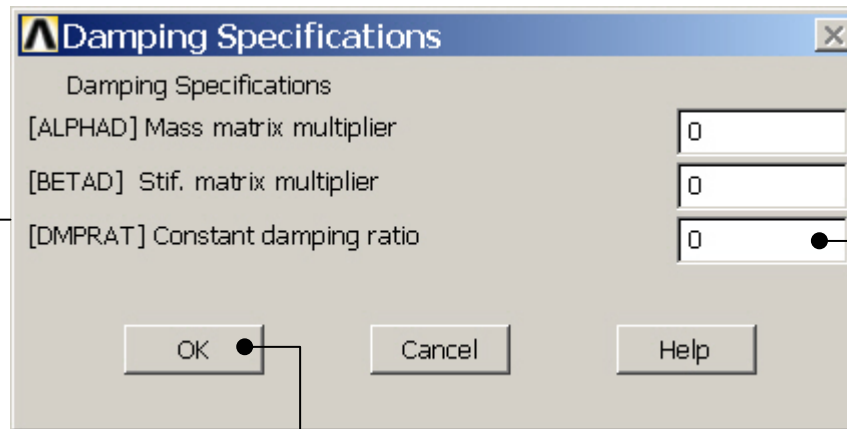
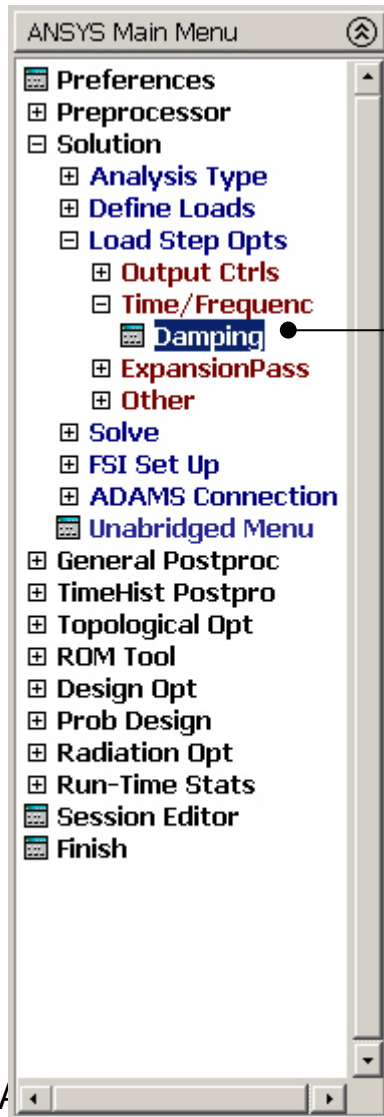
Press OK

Example – Define Loads

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



Example - Damping

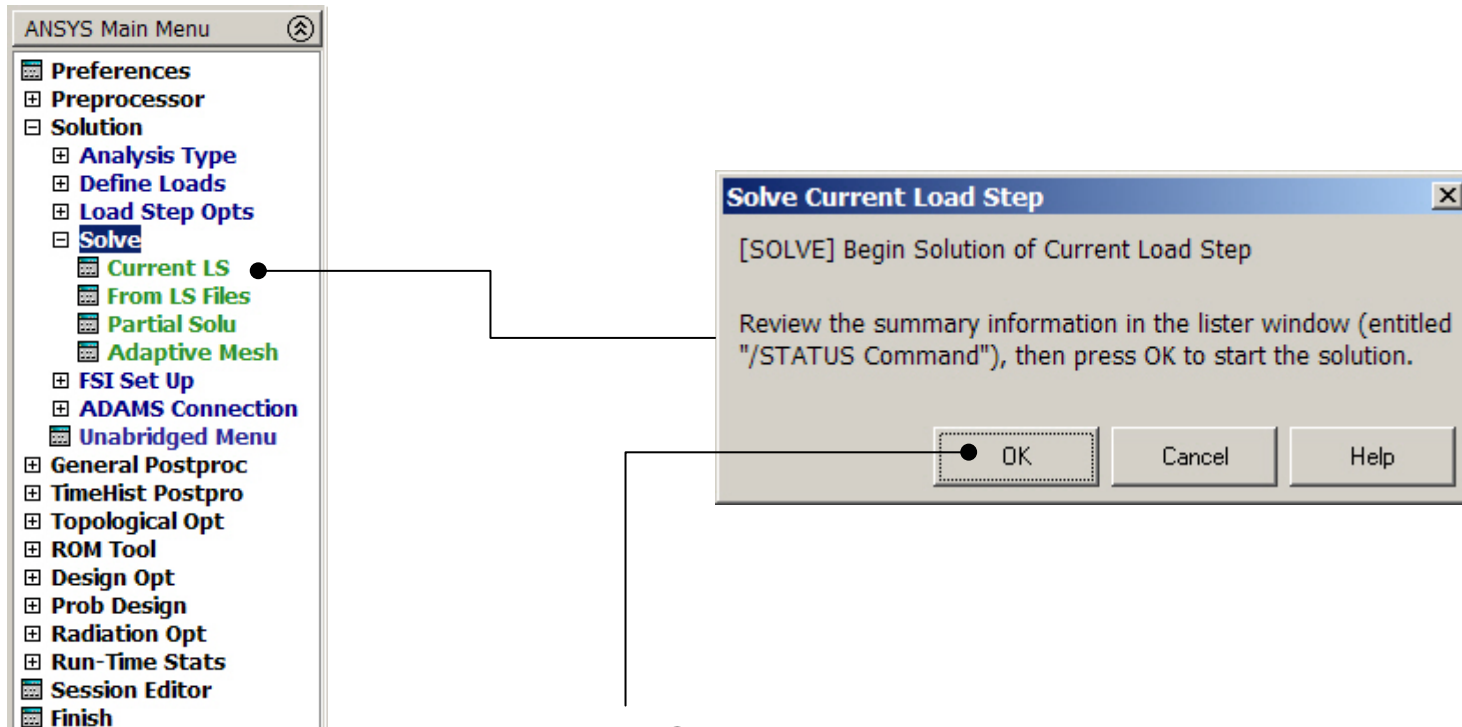


Press OK

Enter 0.02 for Constant damping ratio

Example - Solve

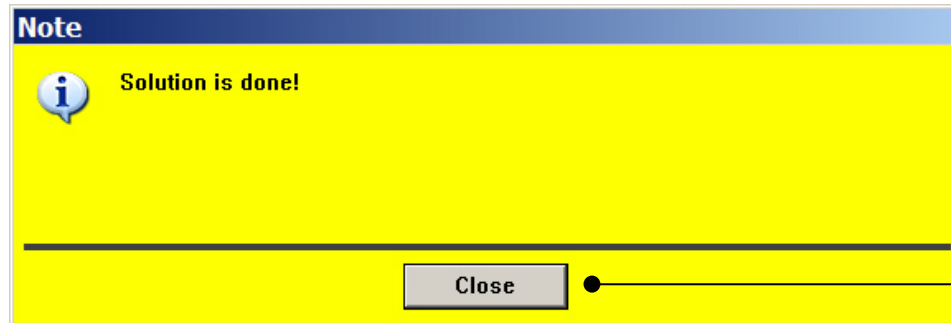
Solution > Solve > Current LS



Press OK

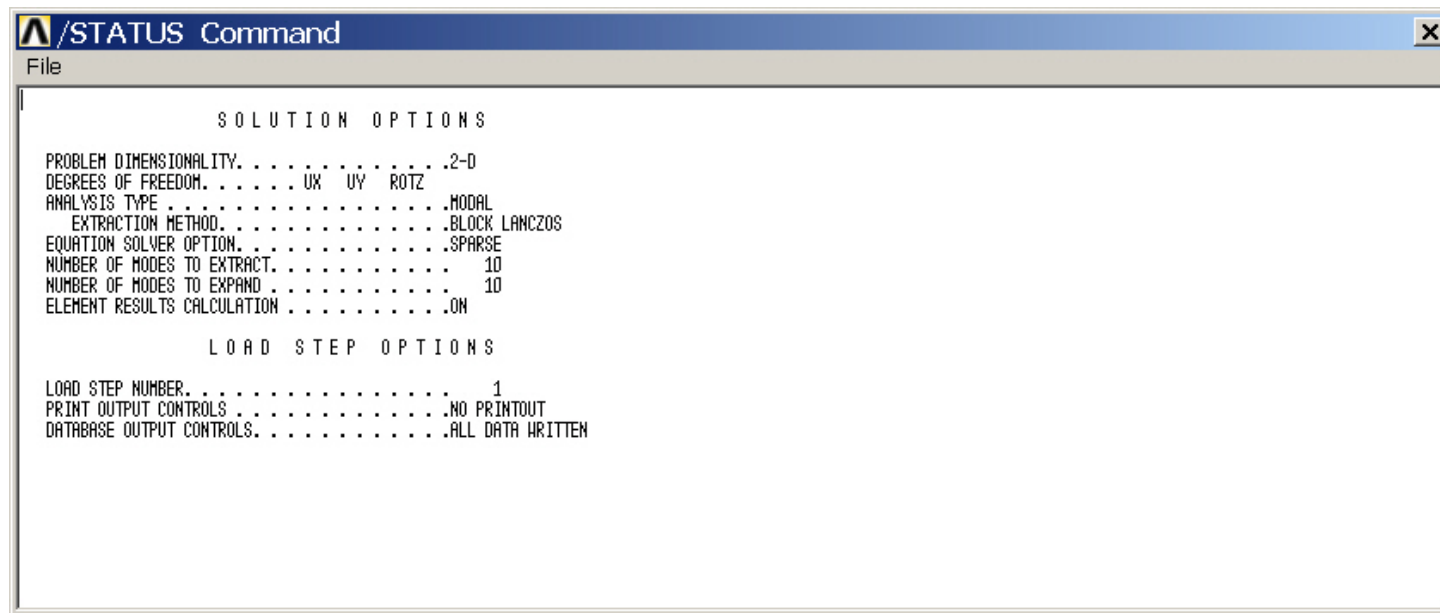
Example0410

Example - Solve



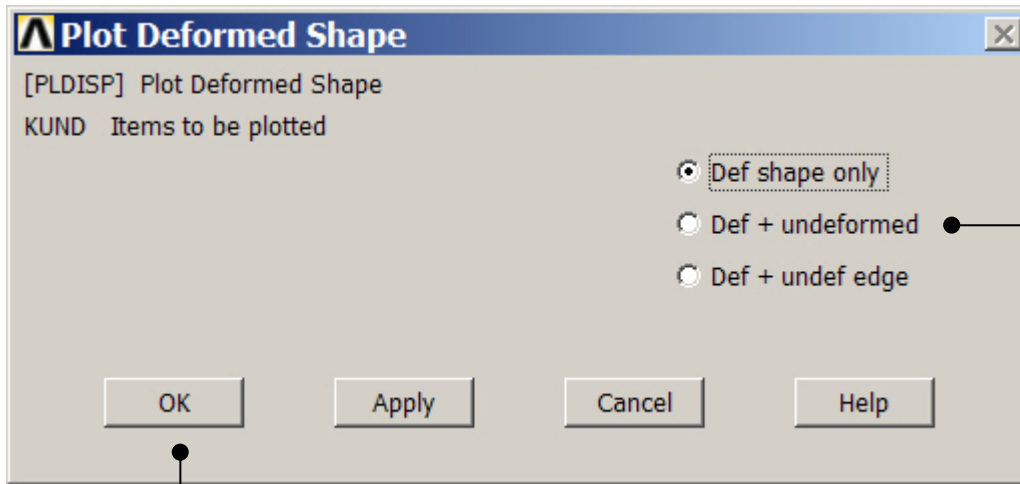
Press Close

Press here
to Close



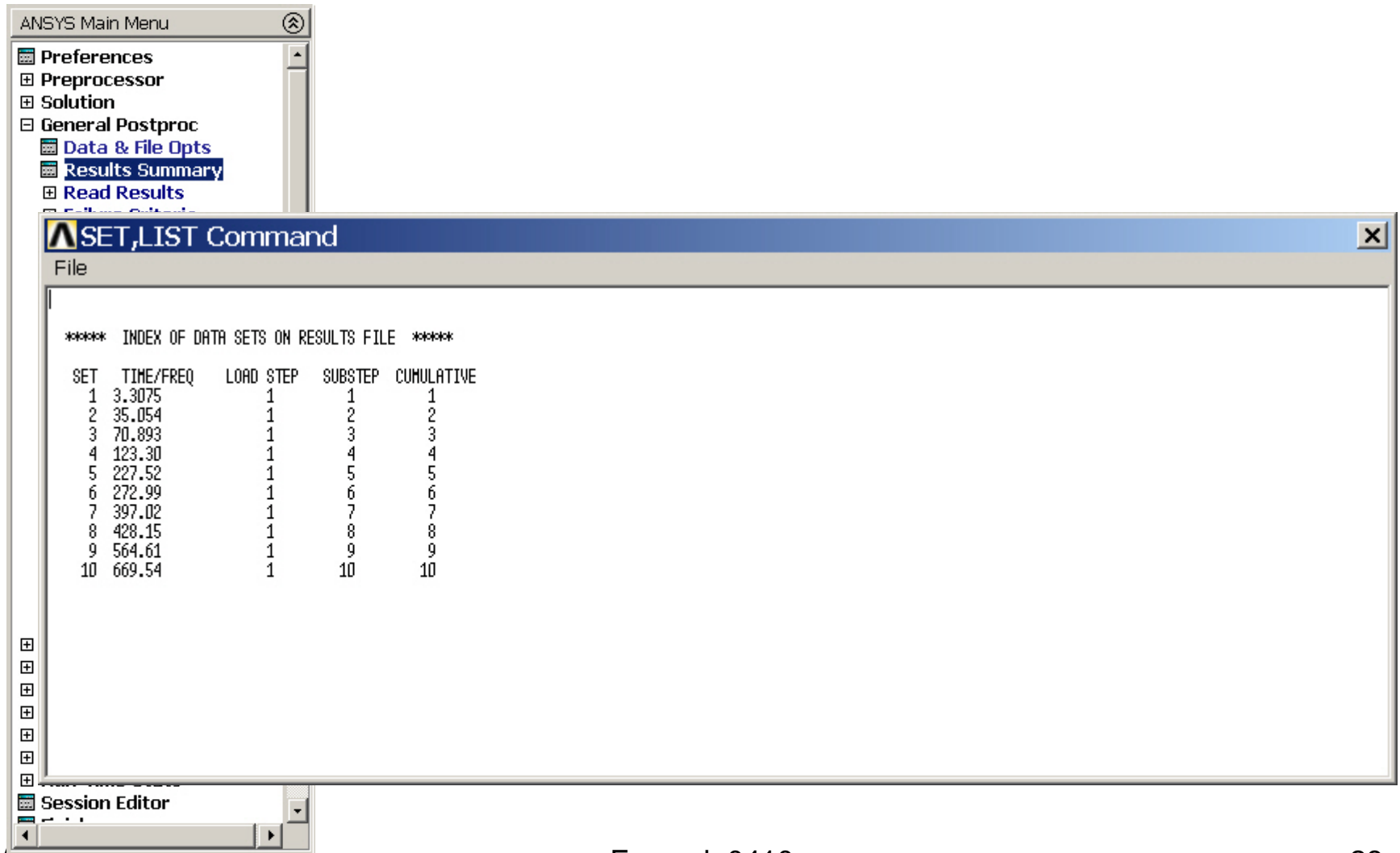
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



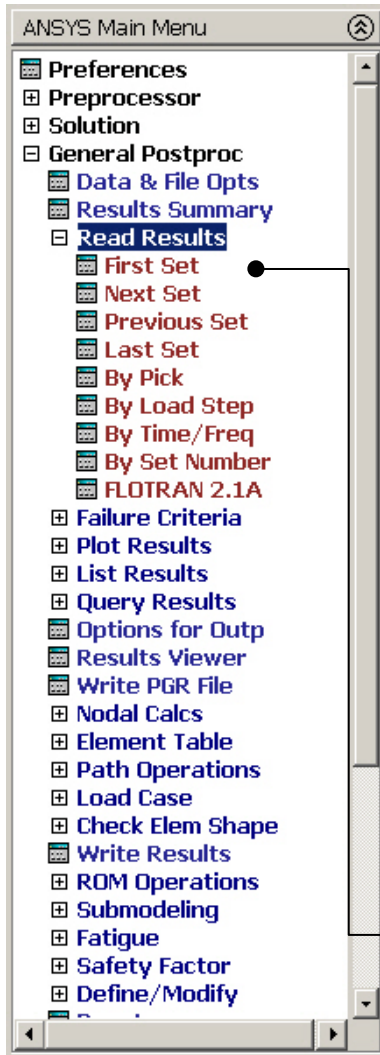
Select "Def+undeformed"
and Press OK

Example – Results Summary

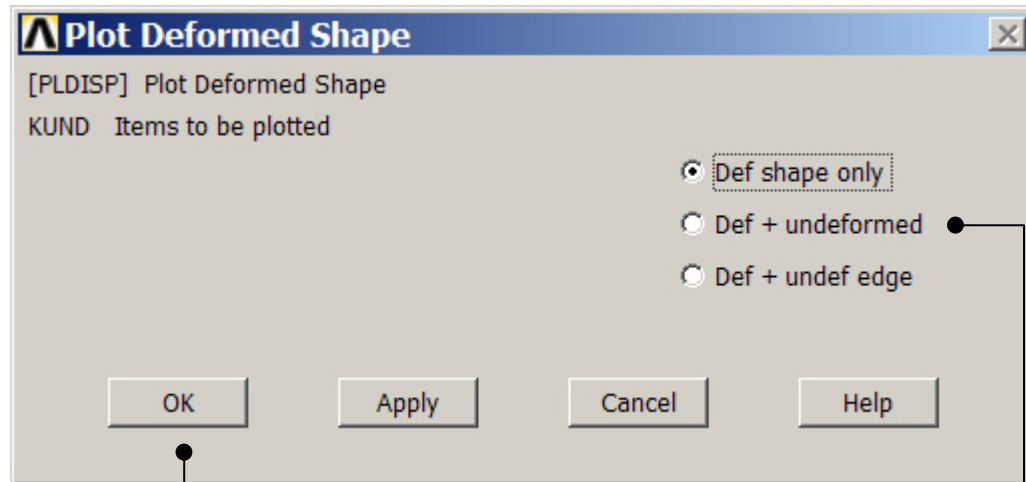


Example – Read Results

General Postproc > Plot Results > Deformed Shape



Read first set

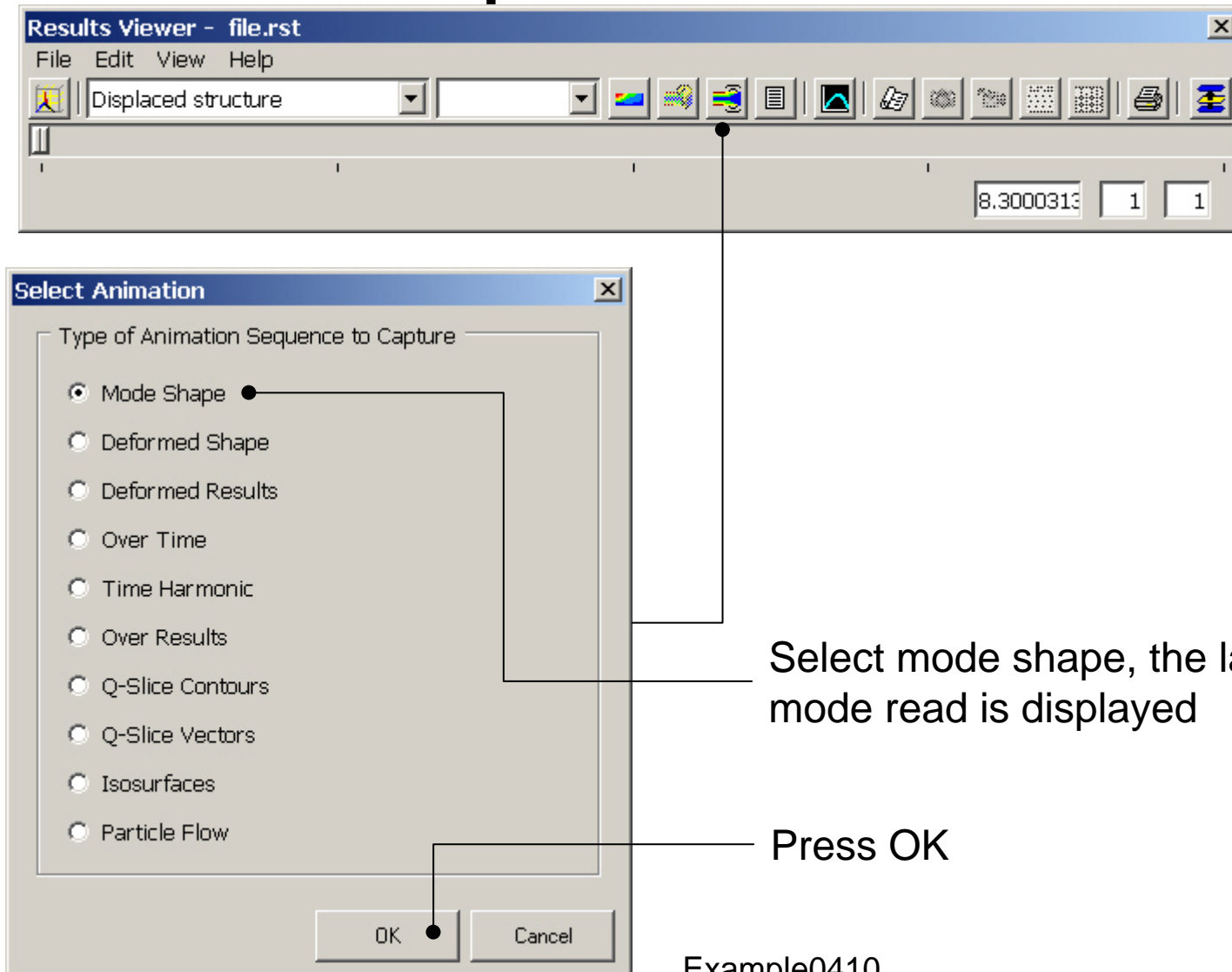


Select "Def+undeformed"
and Press OK

Example – Mode 1



Example – Result viewer

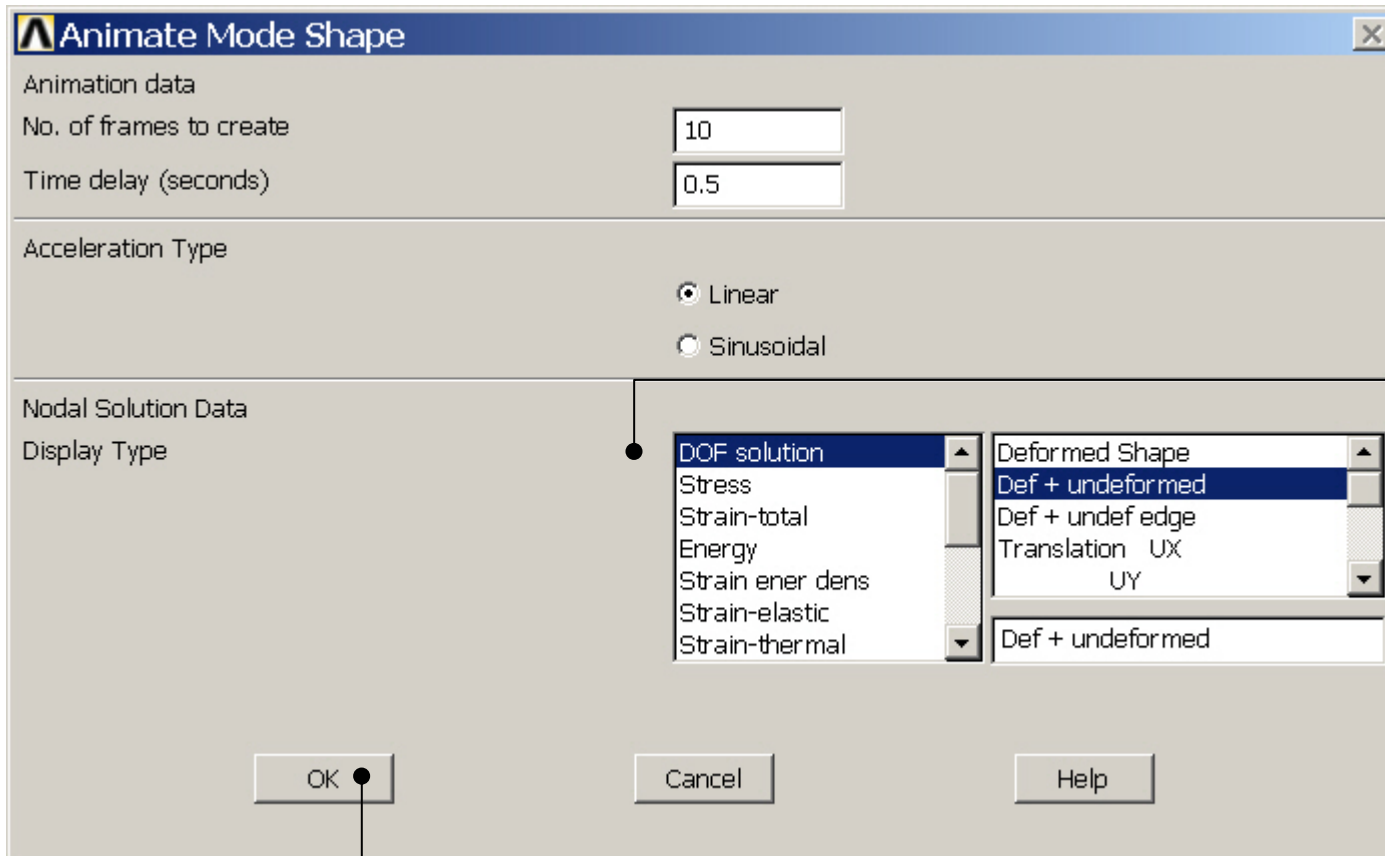


Select mode shape, the last mode read is displayed

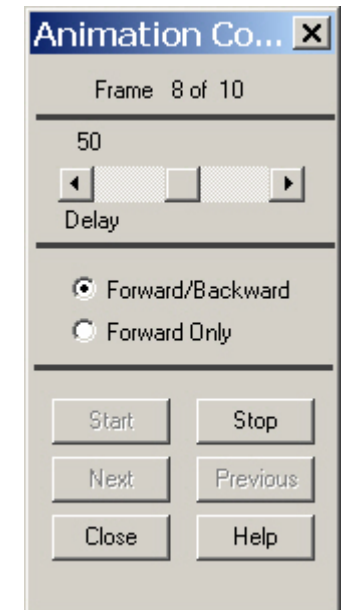
Press OK

Example0410

Example – Result viewer

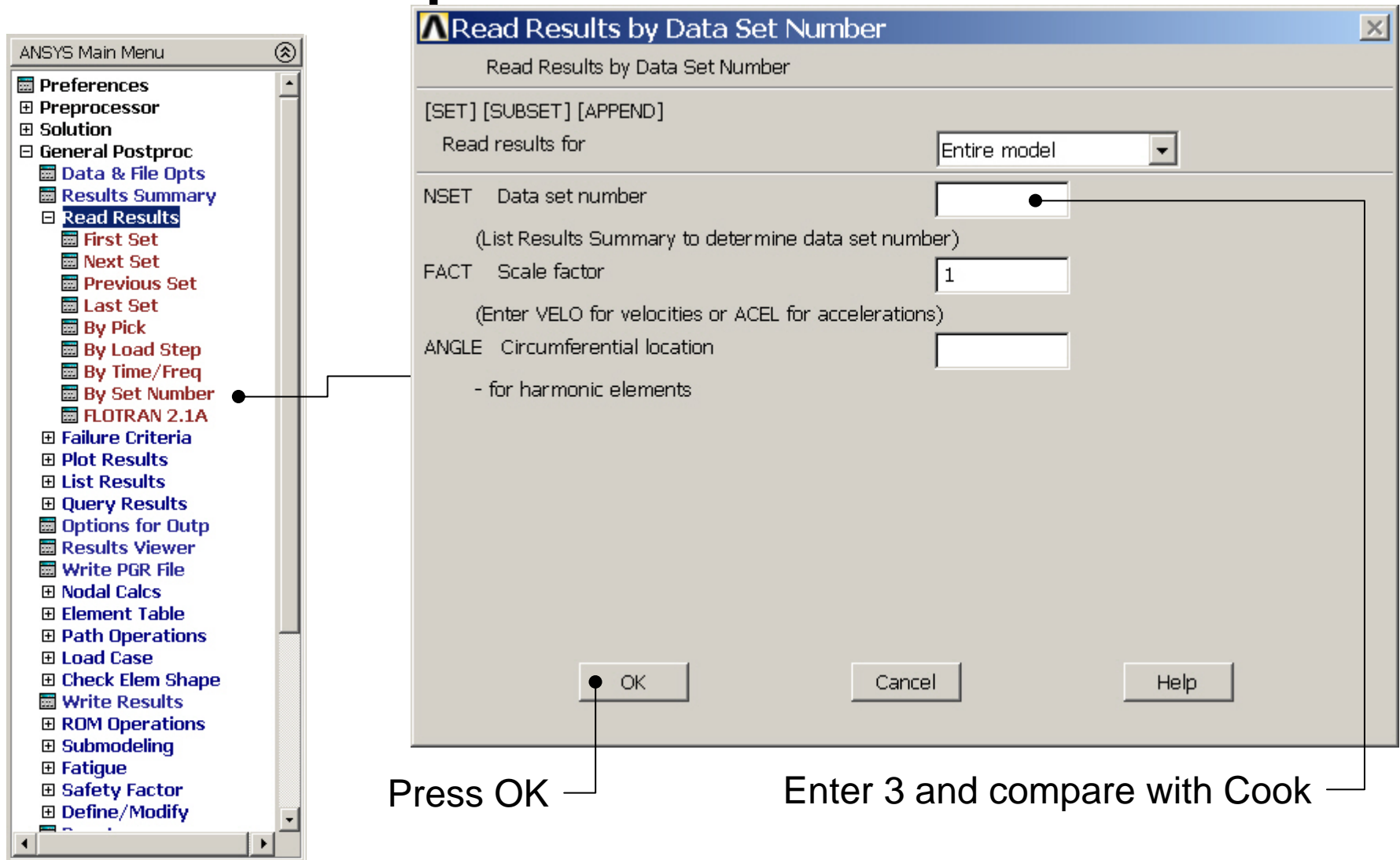


Select DOF solution and Def+undeformed

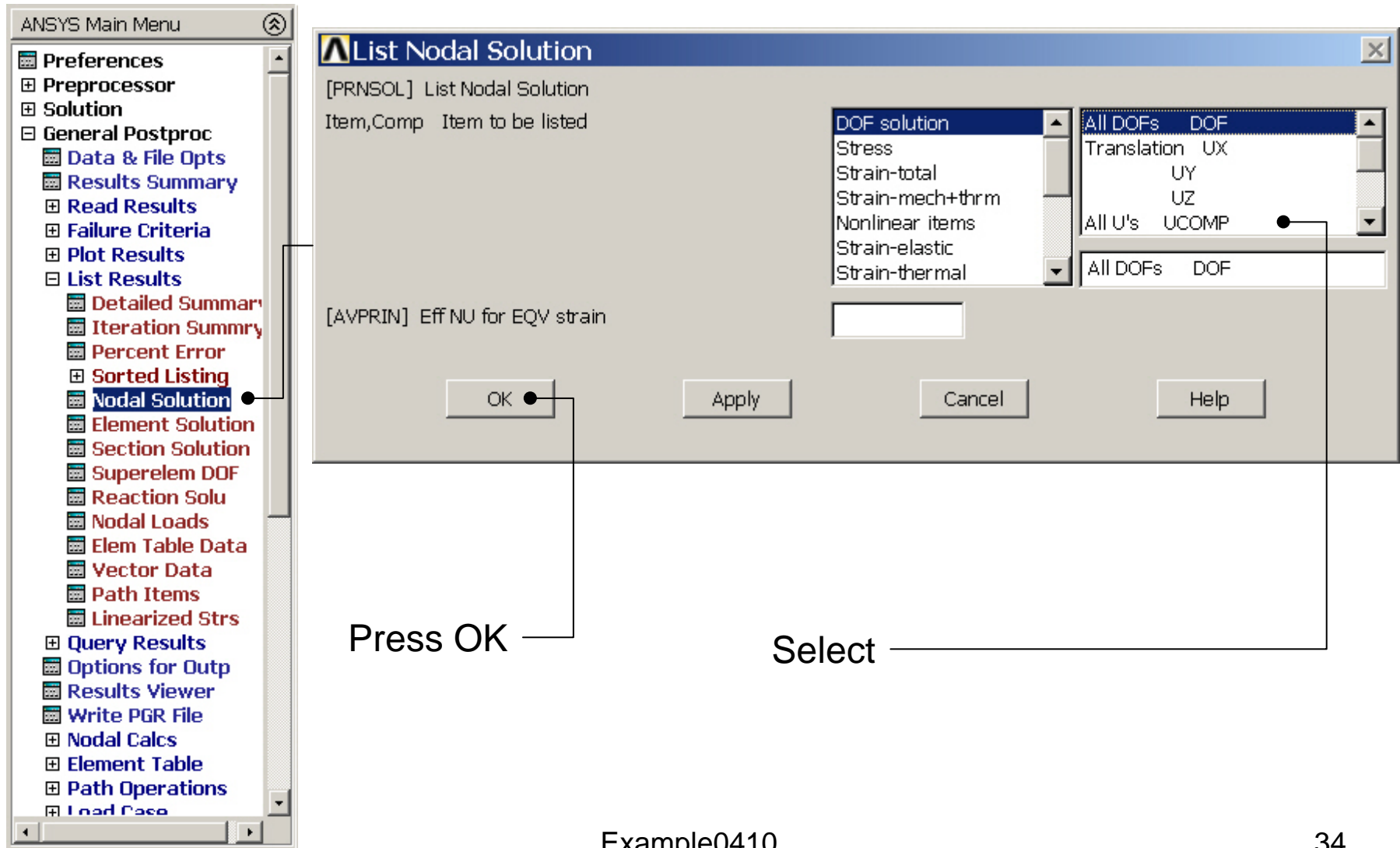


Press OK and animation is controlled by the following dialog

Example – Read Results



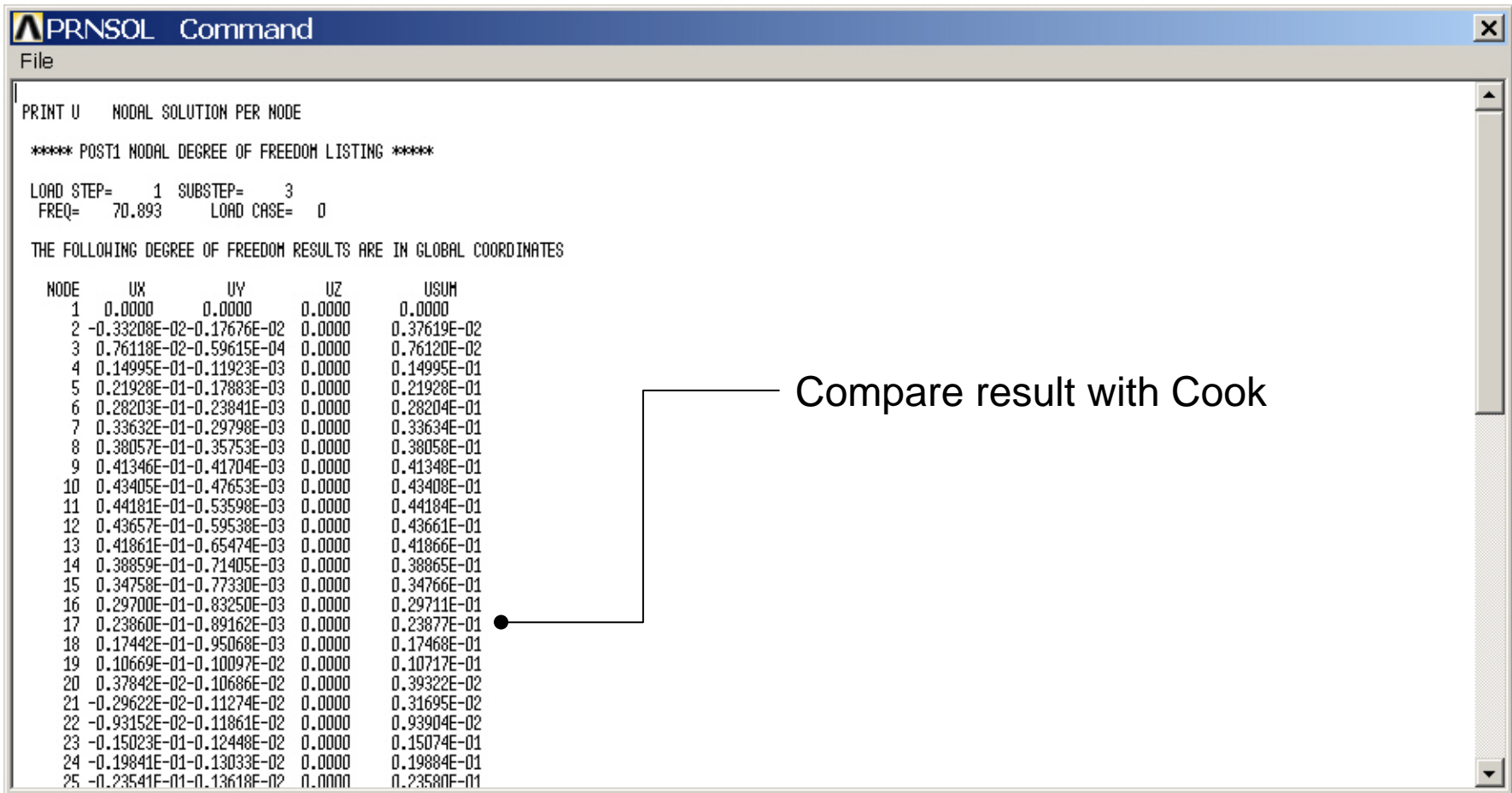
Example - List



Example0410

34

Example - List



```
PRNSOL Command
File

PRINT U    NODAL SOLUTION PER NODE

**** POST1 NODAL DEGREE OF FREEDOM LISTING ****

LOAD STEP= 1 SUBSTEP= 3
FREQ= 70.893 LOAD CASE= 0

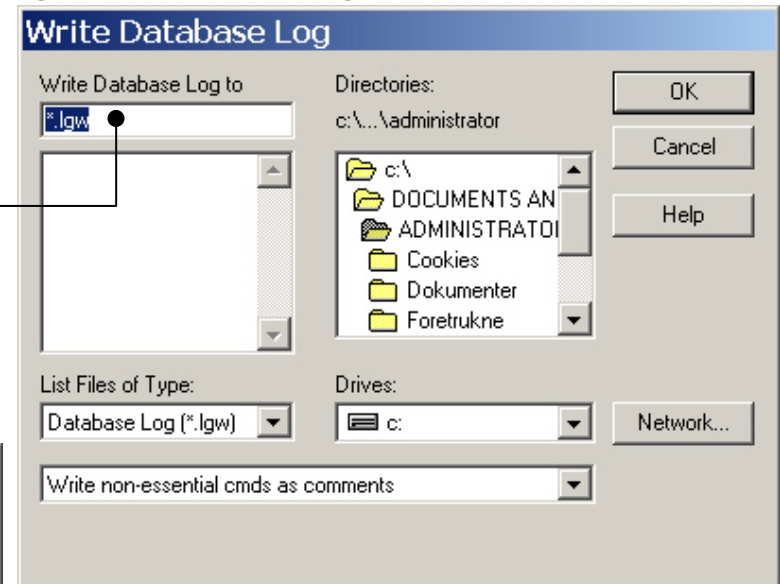
THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN GLOBAL COORDINATES

  NODE      UX      UY      UZ      USUM
  1      0.0000      0.0000      0.0000      0.0000
  2 -0.33208E-02-0.17676E-02      0.0000      0.37619E-02
  3  0.76118E-02-0.59615E-04      0.0000      0.76120E-02
  4  0.14995E-01-0.11923E-03      0.0000      0.14995E-01
  5  0.21928E-01-0.17883E-03      0.0000      0.21928E-01
  6  0.28203E-01-0.23841E-03      0.0000      0.28204E-01
  7  0.33632E-01-0.29798E-03      0.0000      0.33634E-01
  8  0.38057E-01-0.35753E-03      0.0000      0.38058E-01
  9  0.41346E-01-0.41704E-03      0.0000      0.41348E-01
 10  0.43405E-01-0.47653E-03      0.0000      0.43408E-01
 11  0.44181E-01-0.53598E-03      0.0000      0.44184E-01
 12  0.43657E-01-0.59538E-03      0.0000      0.43661E-01
 13  0.41861E-01-0.65474E-03      0.0000      0.41866E-01
 14  0.38859E-01-0.71405E-03      0.0000      0.38865E-01
 15  0.34758E-01-0.77330E-03      0.0000      0.34766E-01
 16  0.29700E-01-0.83250E-03      0.0000      0.29711E-01
 17  0.23860E-01-0.89162E-03      0.0000      0.23877E-01
 18  0.17442E-01-0.95068E-03      0.0000      0.17468E-01
 19  0.10669E-01-0.10097E-02      0.0000      0.10717E-01
 20  0.37842E-02-0.10686E-02      0.0000      0.39322E-02
 21 -0.29622E-02-0.11274E-02      0.0000      0.31695E-02
 22 -0.93152E-02-0.11861E-02      0.0000      0.93904E-02
 23 -0.15023E-01-0.12448E-02      0.0000      0.15074E-01
 24 -0.19841E-01-0.13033E-02      0.0000      0.19884E-01
 25 -0.23541E-01-0.13618E-02      0.0000      0.23580E-01
```

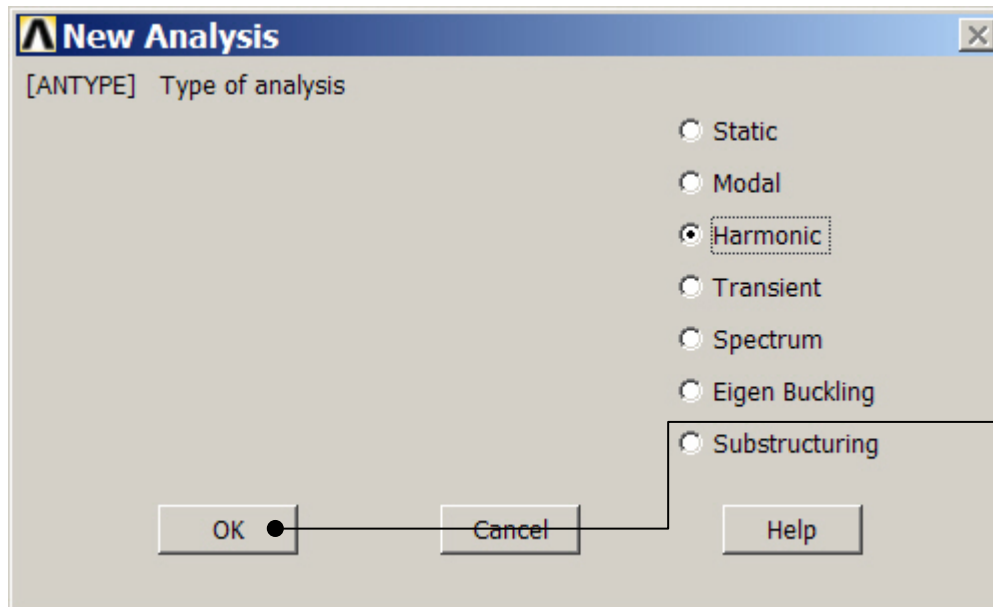
Compare result with Cook

Example – Analysis Type

File > Write DB log file
Enter “example0410.lgw”

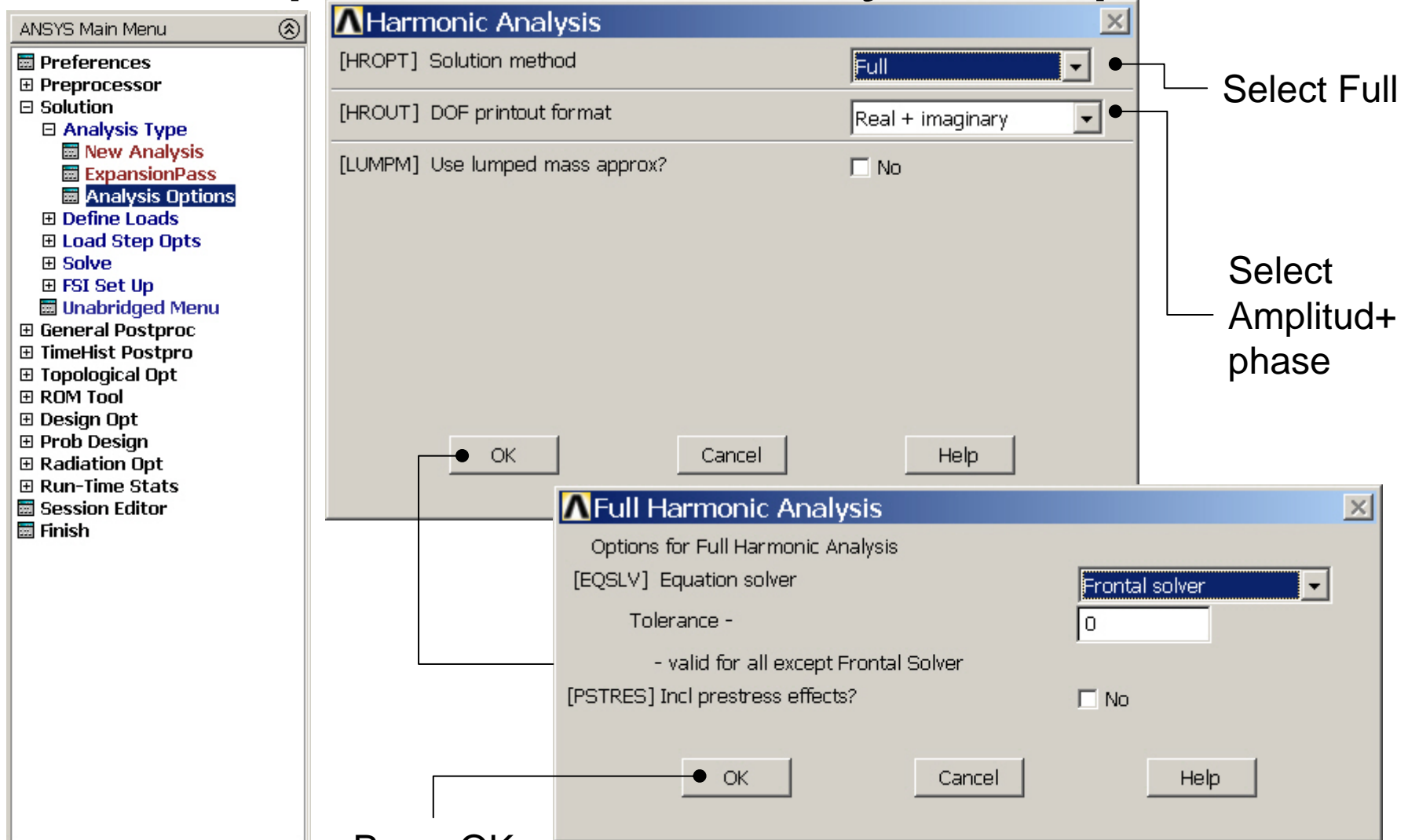


Solution > Analysis Type > New Analysis



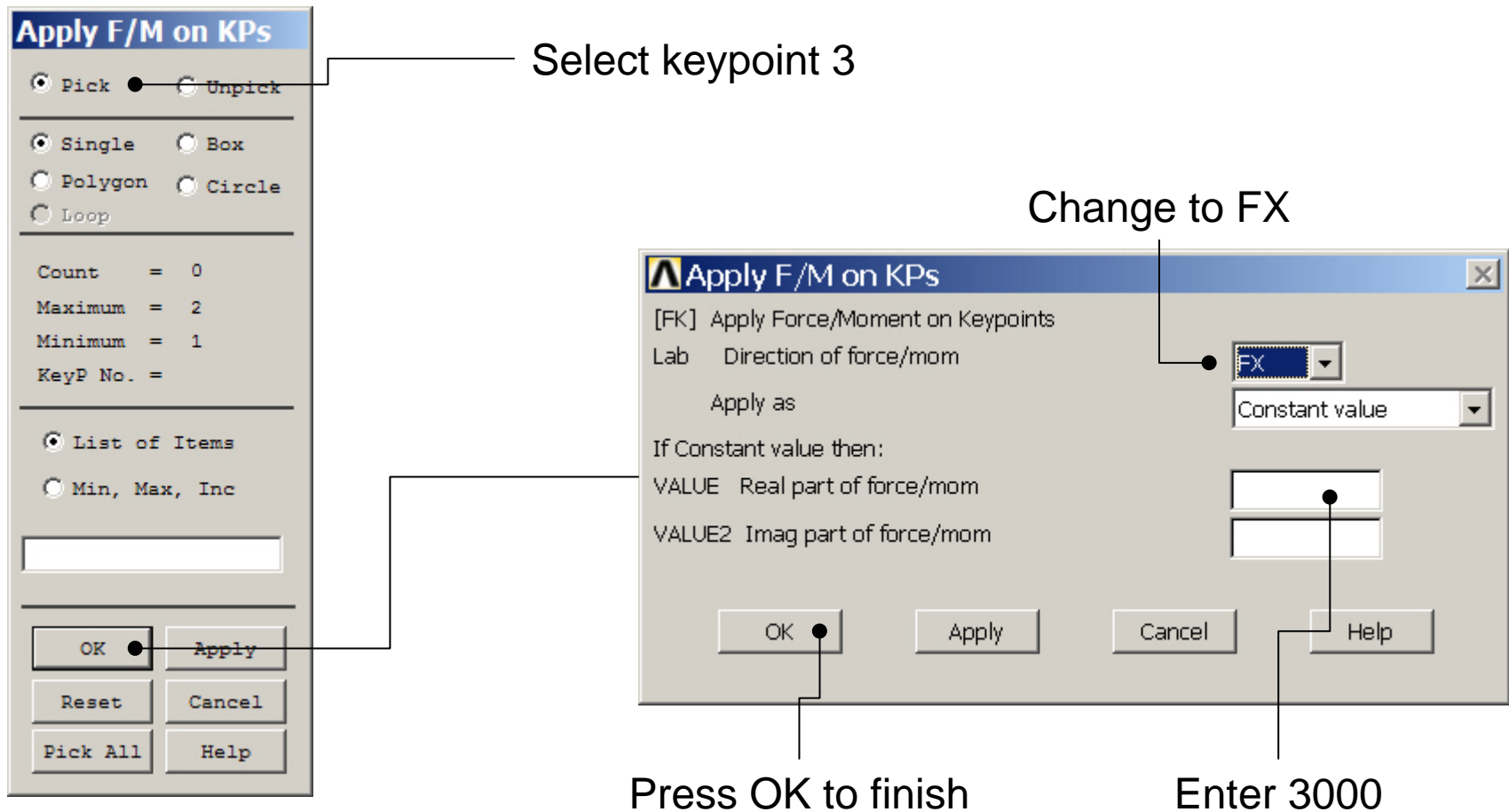
Press OK

Example – HRA Analysis Options

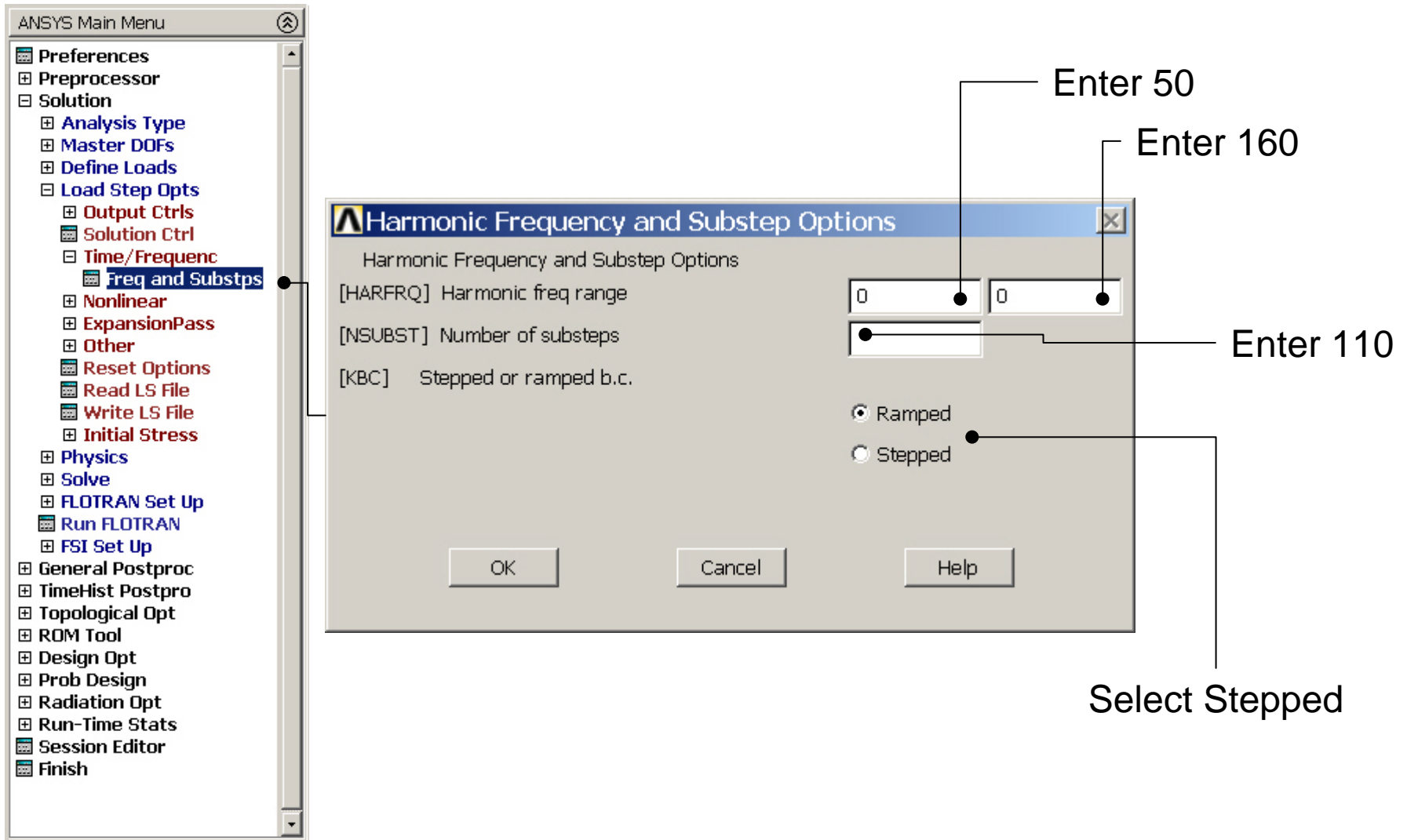


Example – Define Loads

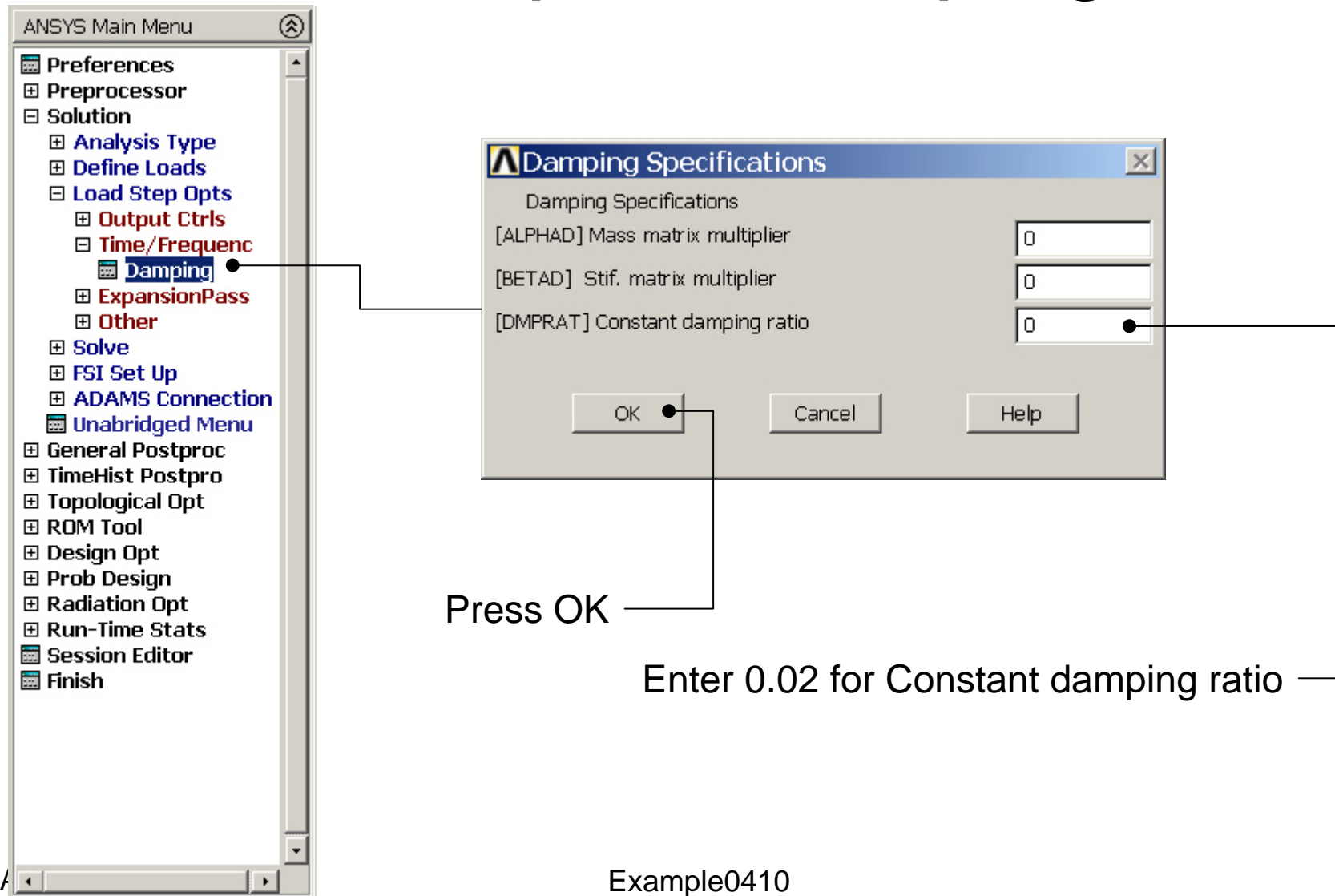
Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints



Example - Freq and Substps



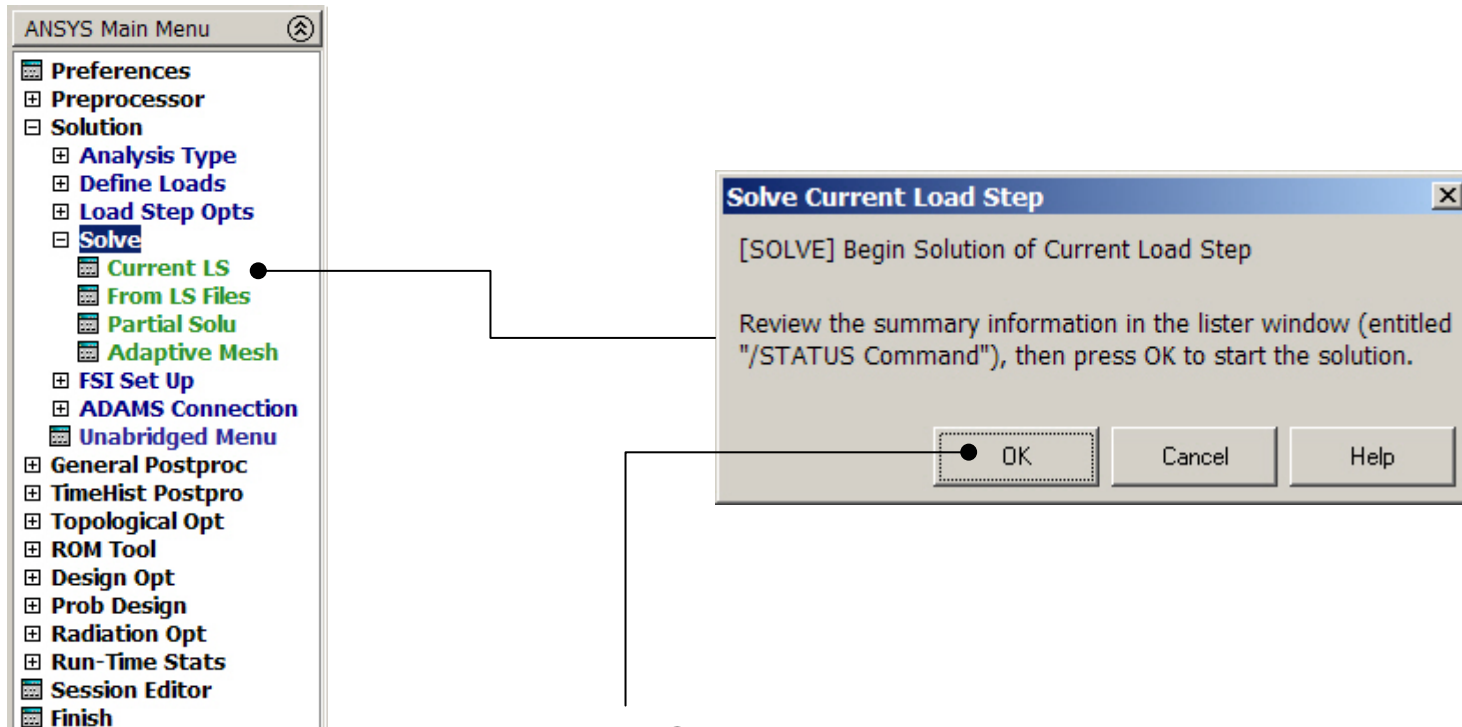
Example - Damping



Example0410

Example - Solve

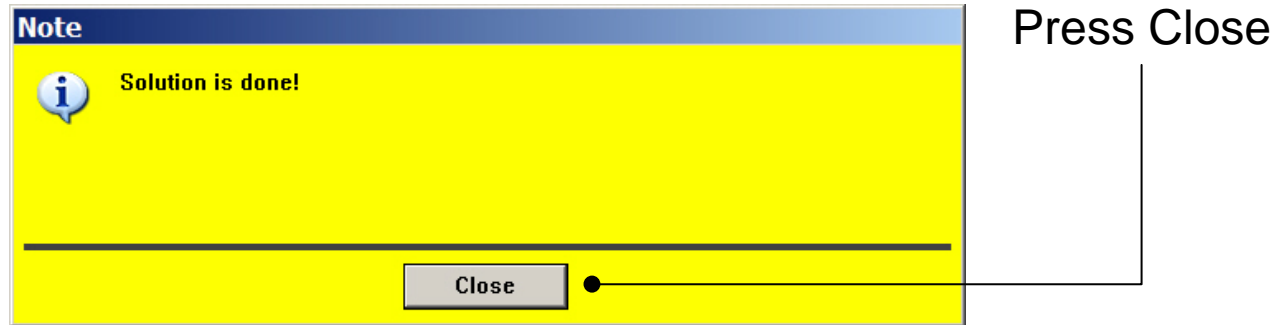
Solution > Solve > Current LS



Press OK

Example0410

Example - Solve



Example – TimeHistory Postpro

The image shows the ANSYS Main Menu on the left and the Time History Variables dialog box on the right. The Main Menu includes options like Preferences, Preprocessor, Solution, General Postproc, TimeHist Postpro, and Finish. The Time History Variables dialog box has a title bar 'Time History Variables - file.rst' and a menu bar 'File Help'. It contains a 'Variable List' table with columns: Name, Element, Node, Result Item, Minimum, Maximum, X-Axis, and a selection column. The table has one row: 'TIME', 'Time', '1', '1', and 'X-Axis' is selected. Below the table is a 'Calculator' section with a display area and a keypad with various mathematical functions and operators. A line points from the text 'Close this dialog box' to the close button (X) in the top right corner of the dialog box.

ANSYS Main Menu

- Preferences
- Preprocessor
- Solution
- General Postproc
- TimeHist Postpro**
 - Variable Viewer
 - Settings
 - Store Data
 - Define Variables
 - Read LSDYNA Data
 - List Variables
 - List Extremes
 - Graph Variables
 - Math Operations
 - Table Operations
 - Smooth Data
 - Generate Spectrm
 - Reset Postproc
- Topological Opt
- ROM Tool
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

Time History Variables - file.rst

File Help

Variable List

Name	Element	Node	Result Item	Minimum	Maximum	X-Axis	
TIME			Time	1	1	X-Axis	

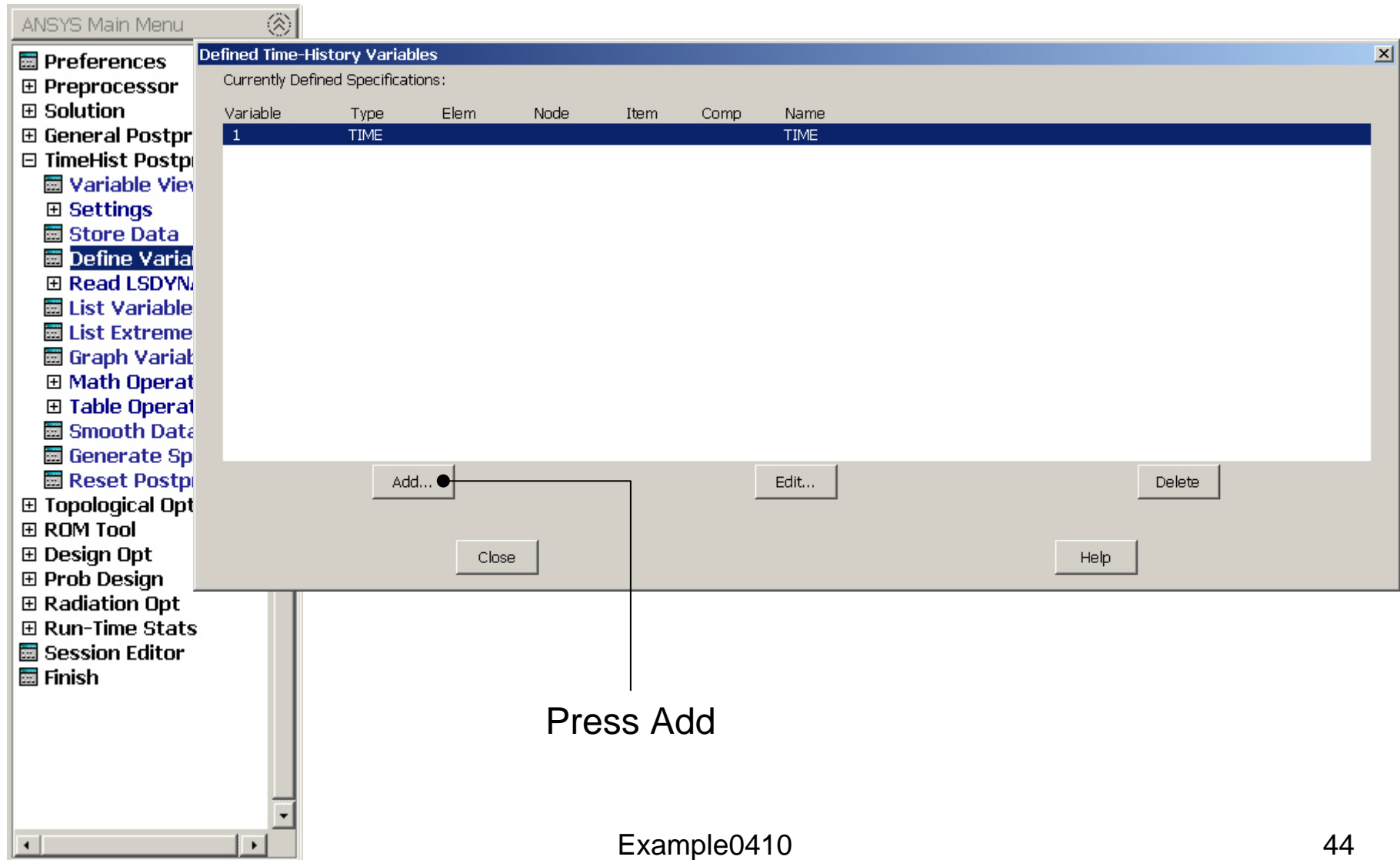
Calculator

Close this dialog box

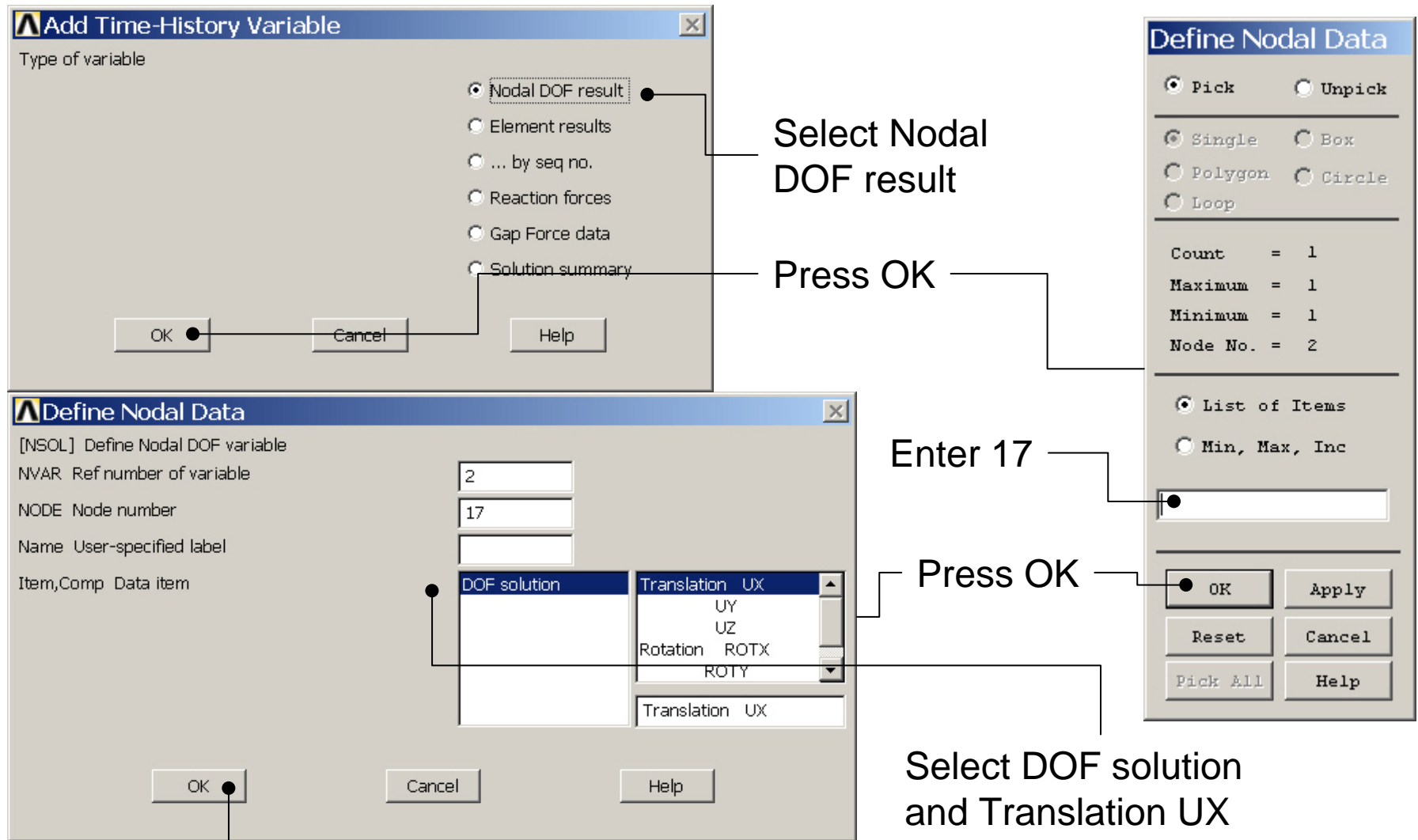
Example0410

43

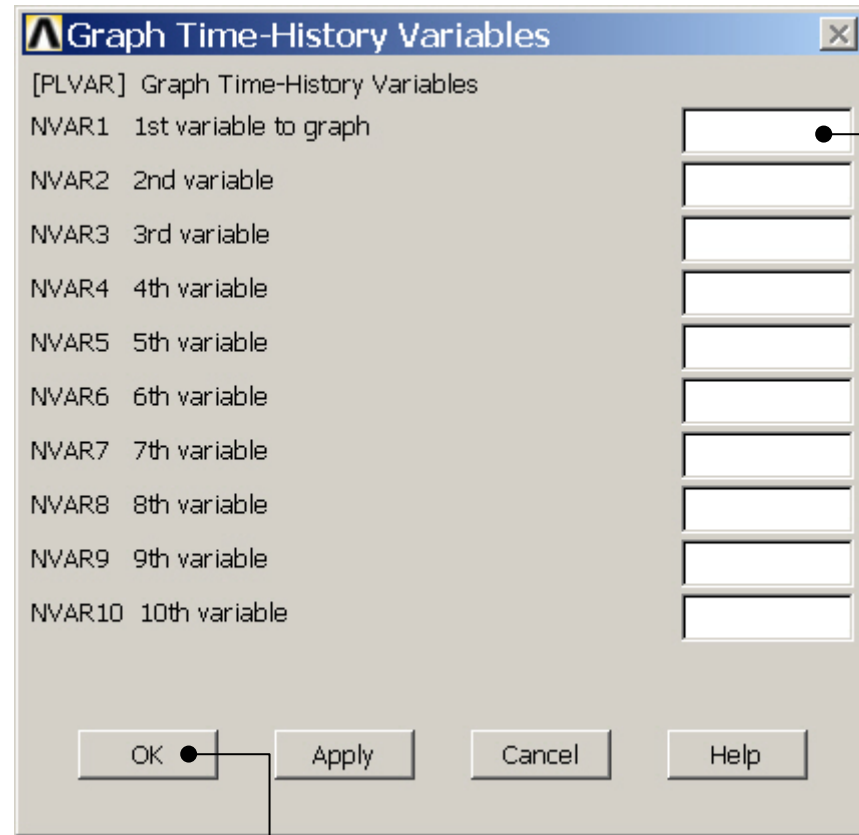
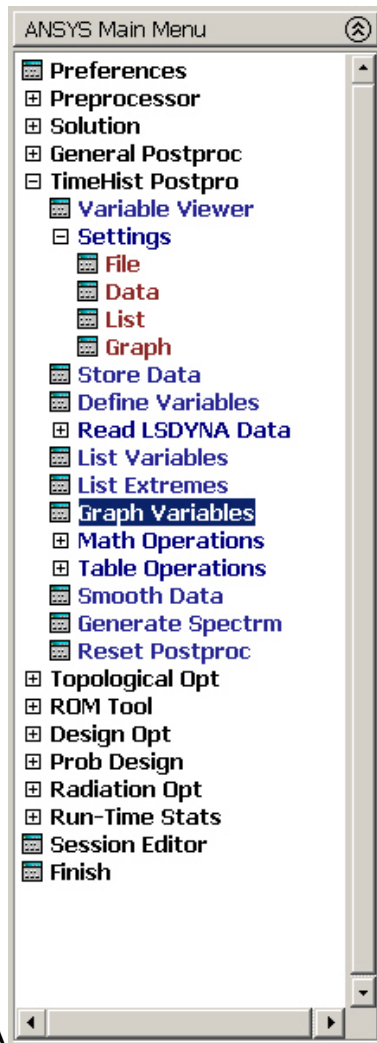
Example – Define Variables



Example – Add Time-History Var.



Example – Graph Variables



Enter 2 to plot
UX for node 17

Press OK

Example0410

Example – Graph Variables

