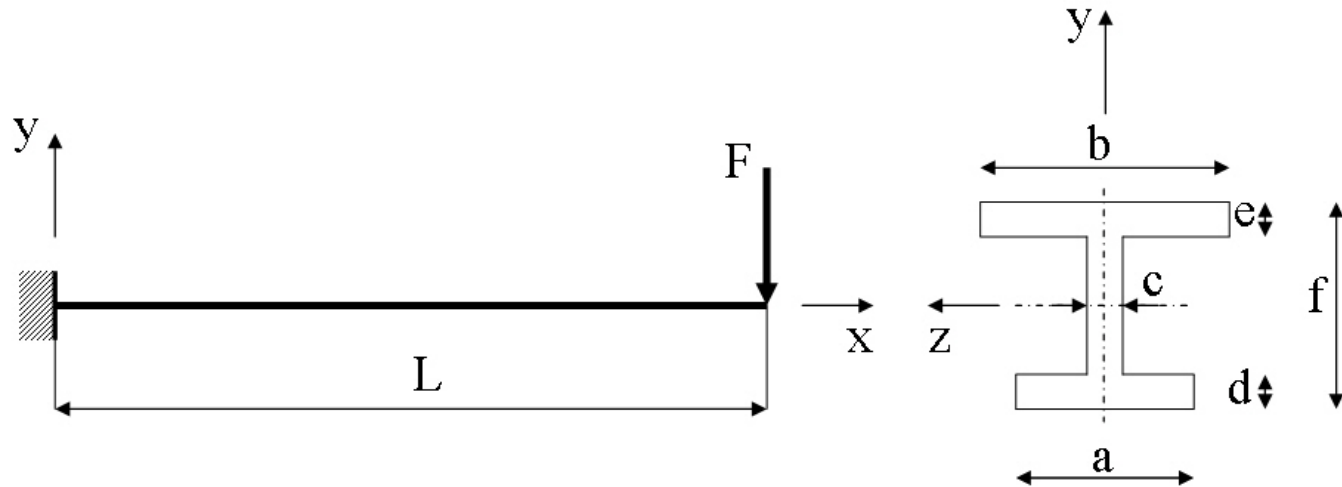


# Course in ANSYS

Example0504

# Example – Cantilever beam



## Objective:

Compute the buckling load

## Tasks:

Display the deflection figure?

## Topics:

Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, saving/restoring, orientation keypoints

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

$$\nu = 0.3$$

$$L = 5000 \text{ mm}$$

$$a = 250 \text{ mm}$$

$$b = 450 \text{ mm}$$

$$c = 10 \text{ mm}$$

$$d = 20 \text{ mm}$$

$$e = 15 \text{ mm}$$

$$f = 350 \text{ mm}$$

$$F = ?$$

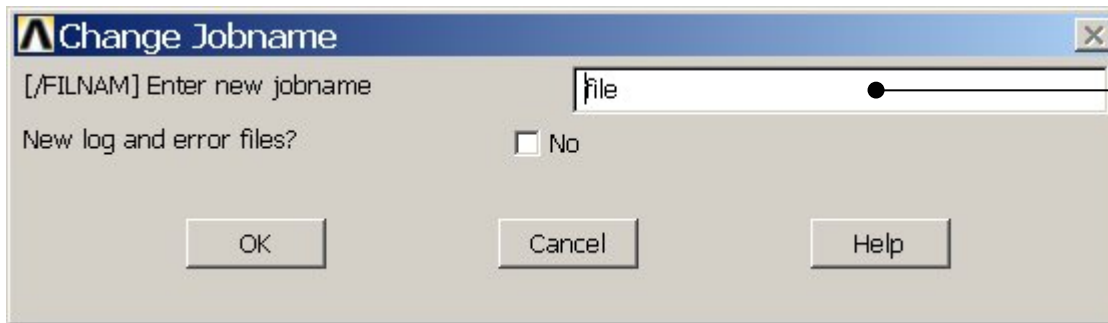
# Example - title

**Utility Menu > File > Change Jobname**

/jobname, Example0504

GUI

Command line entry

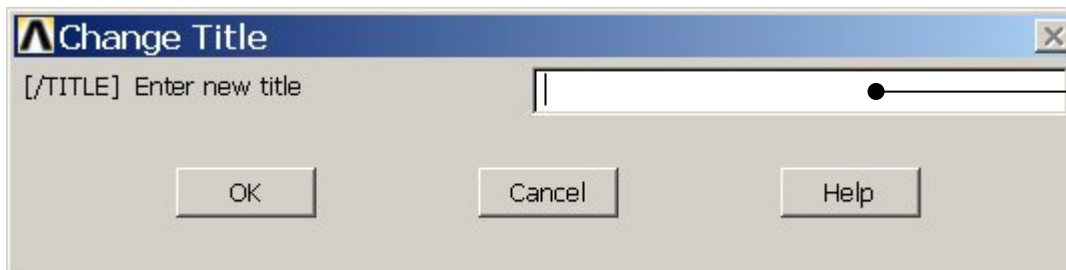


Enter: Example0504

**Utility Menu > File > Change Title**

/title, Cantilever beam

Enter: Cantilever beam



# Example - Keypoints

Note: An empty # result in automatic numbering.

**Preprocessor > Modeling > Create > Keypoints > In Active CS**

/PREP7

K,,,,

K,,5000,,

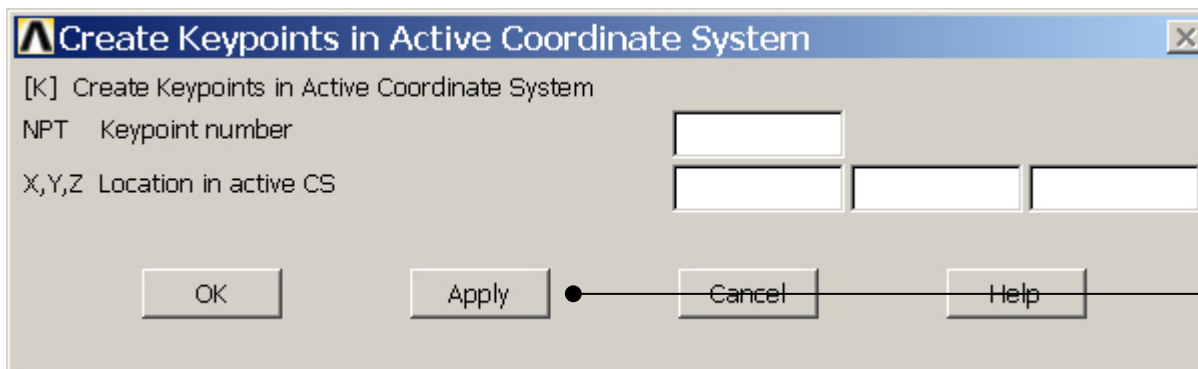
K,,,50,

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 5000,0,0 and  
Press **Apply**  
Enter 0,50,0 and  
Press **Apply**

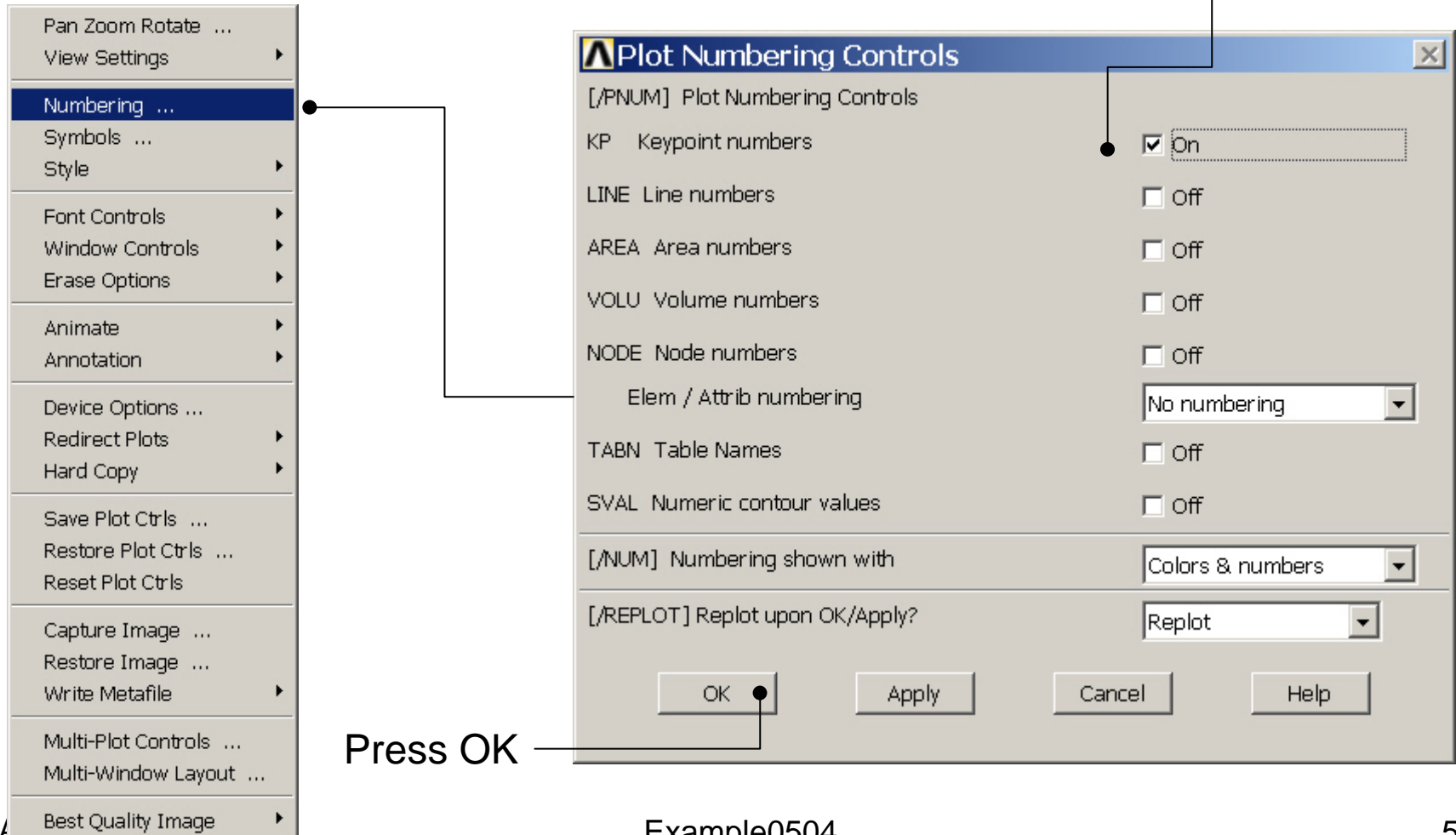


Note: An empty box result in a zero. It is allowed to enter 0.0 in each box.

# Example - Numbering

Utility Menu > PlotCtrls > Numbering

Switch on Keypoint numbers



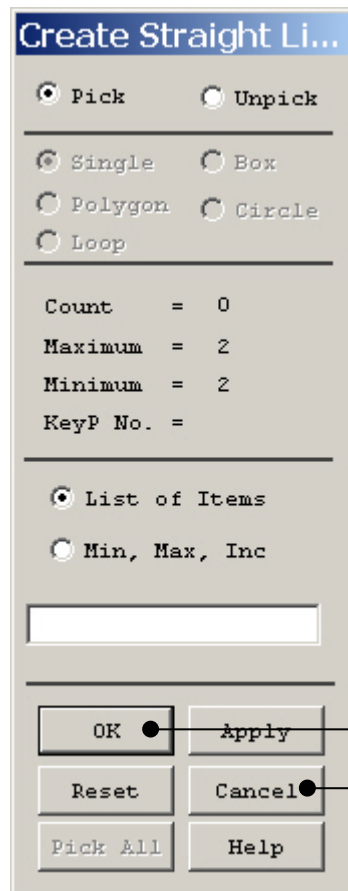
Example0504

# Example - Lines

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

Create a line between Keypoint 1 and Keypoint 2.

L,1,2



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:

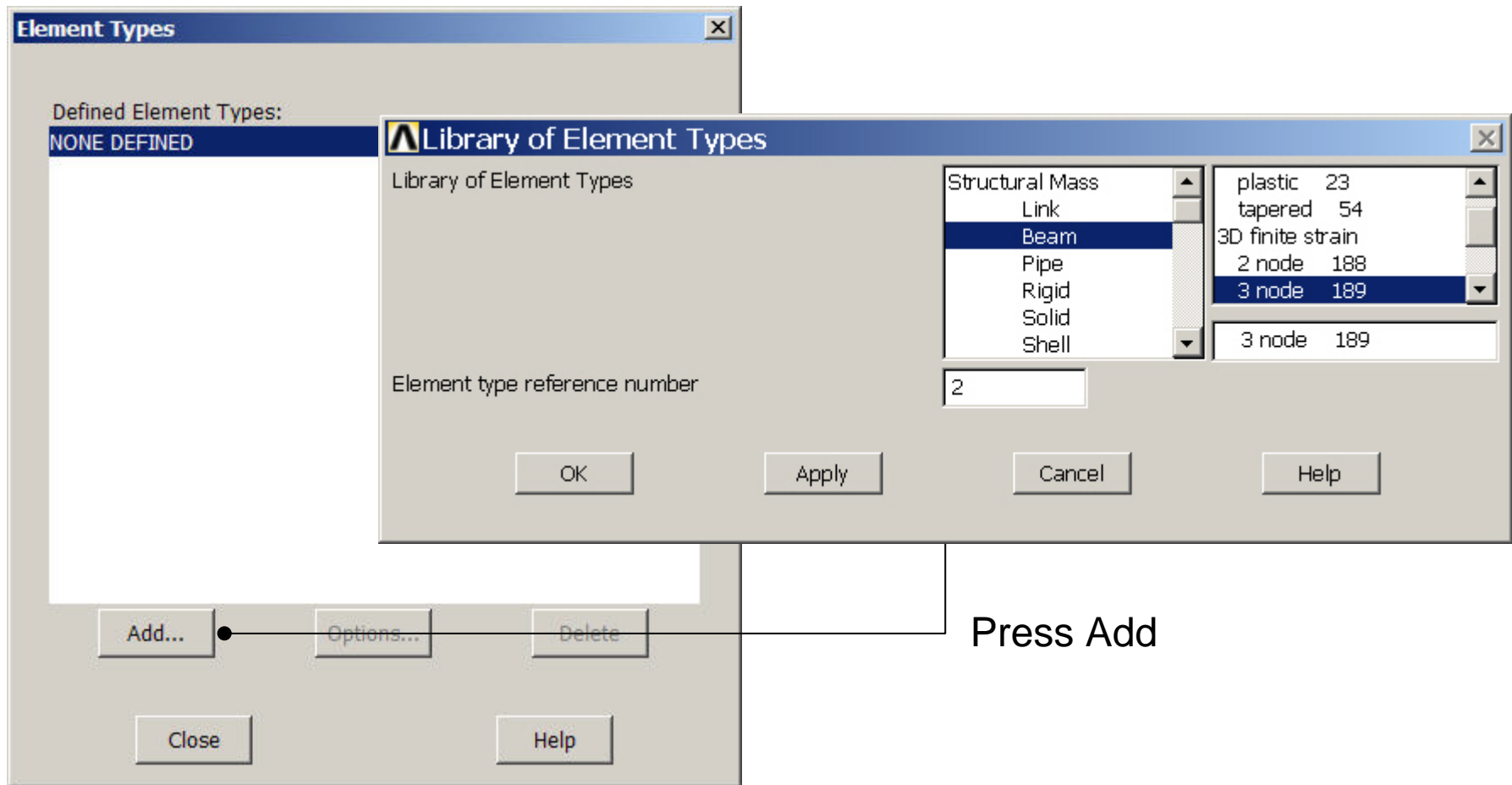
Pick: upward arrow

Unpick: downward arrow

Press OK or Cancel to finish selection

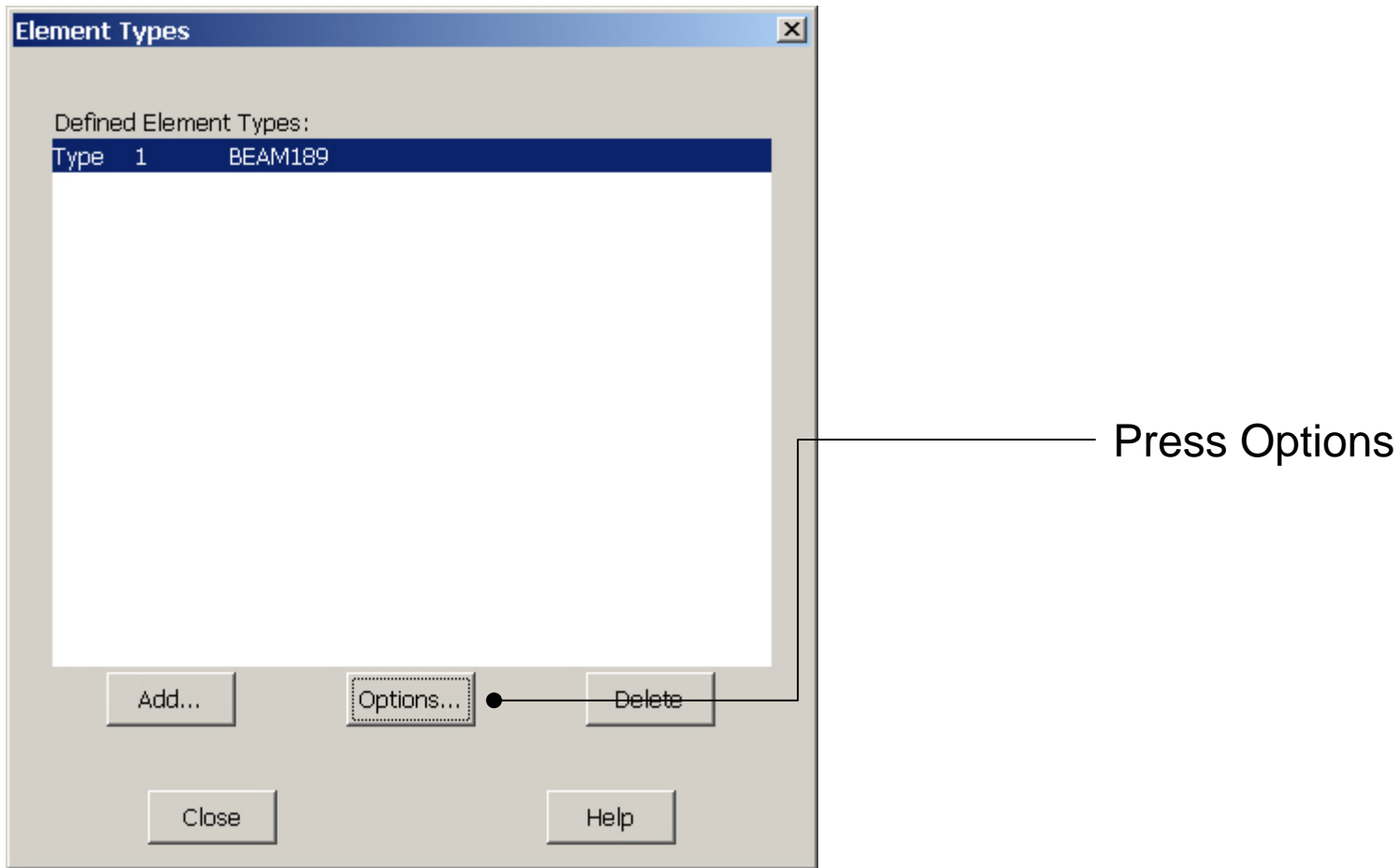
# Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



# Example - Element Type

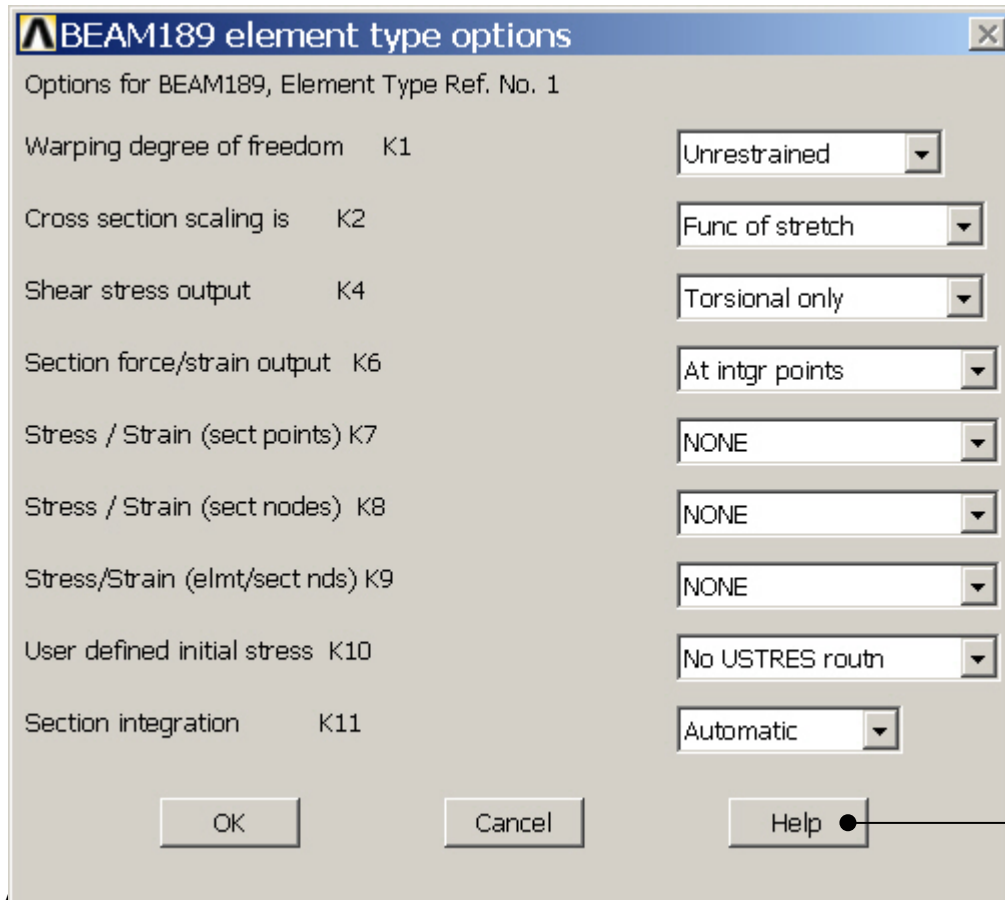
Preprocessor > Element Type > Add/Edit/Delete





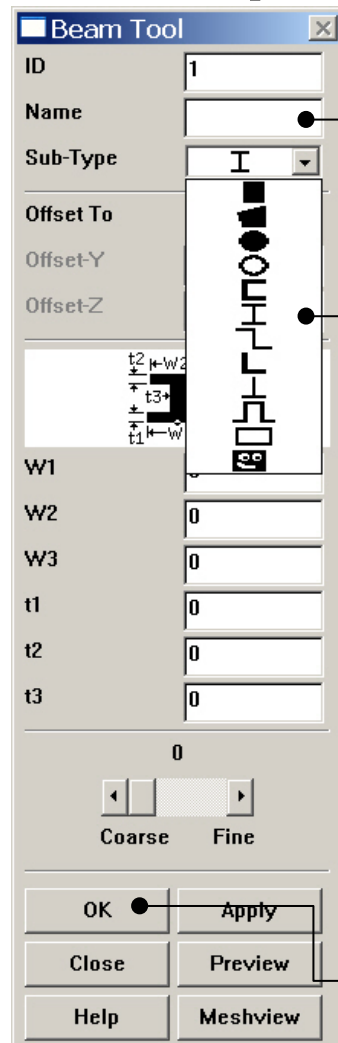
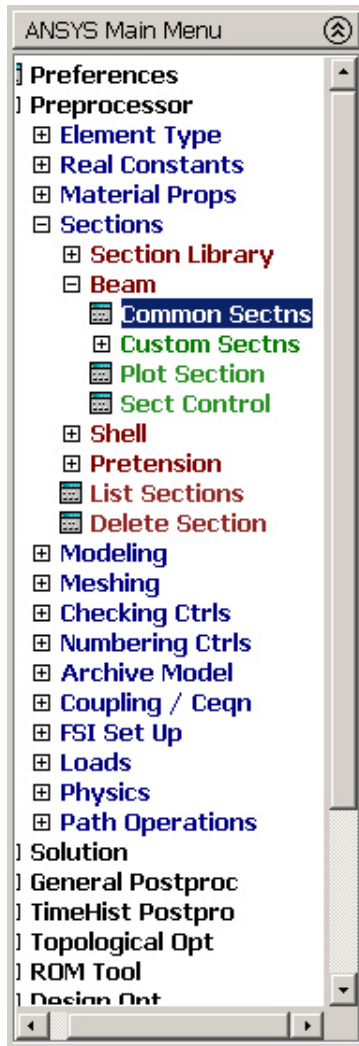
# Example - Element Type

**Preprocessor > Element Type > Add/Edit/Delete**



Press Help to learn more about the element.

# Example - Section

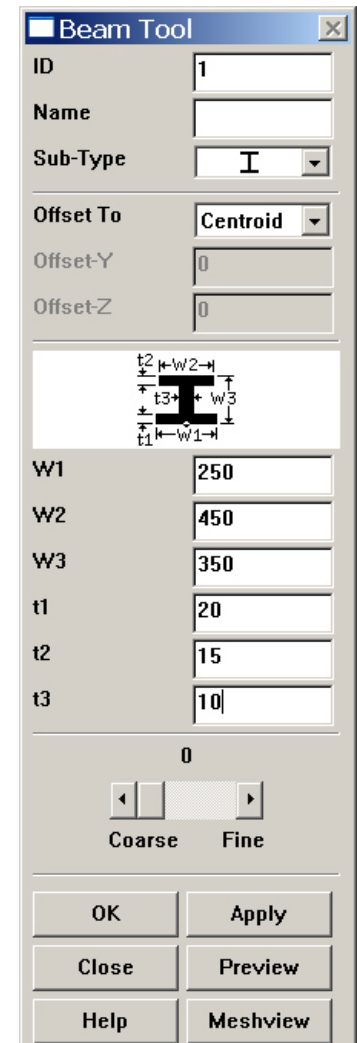


Enter i253

Select the I profile

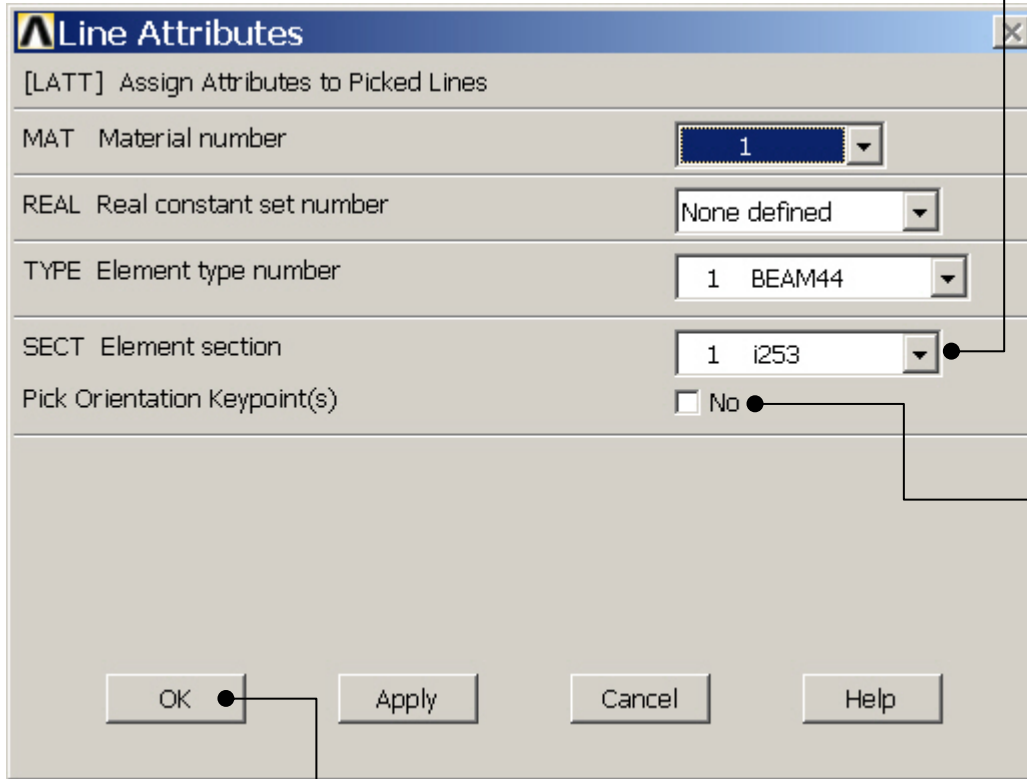
Follow the guidelines at Enter the appropriate cross-sectional data

Press OK to finish



# Example – Line Attributes

Preprocessor > Meshing > Mesh Attributes > Picked Lines

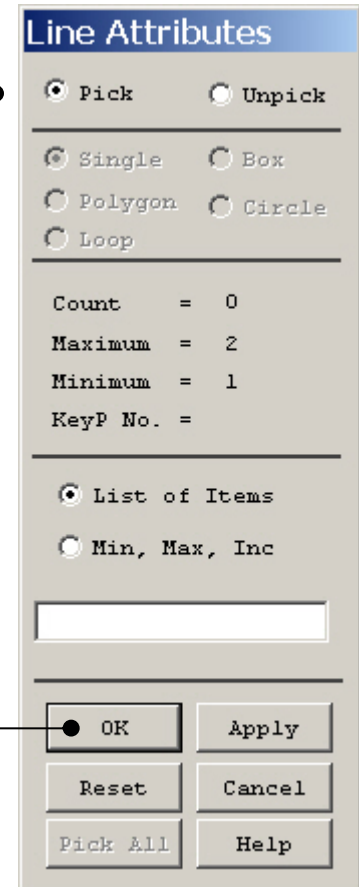


Select section I253

Change to Yes

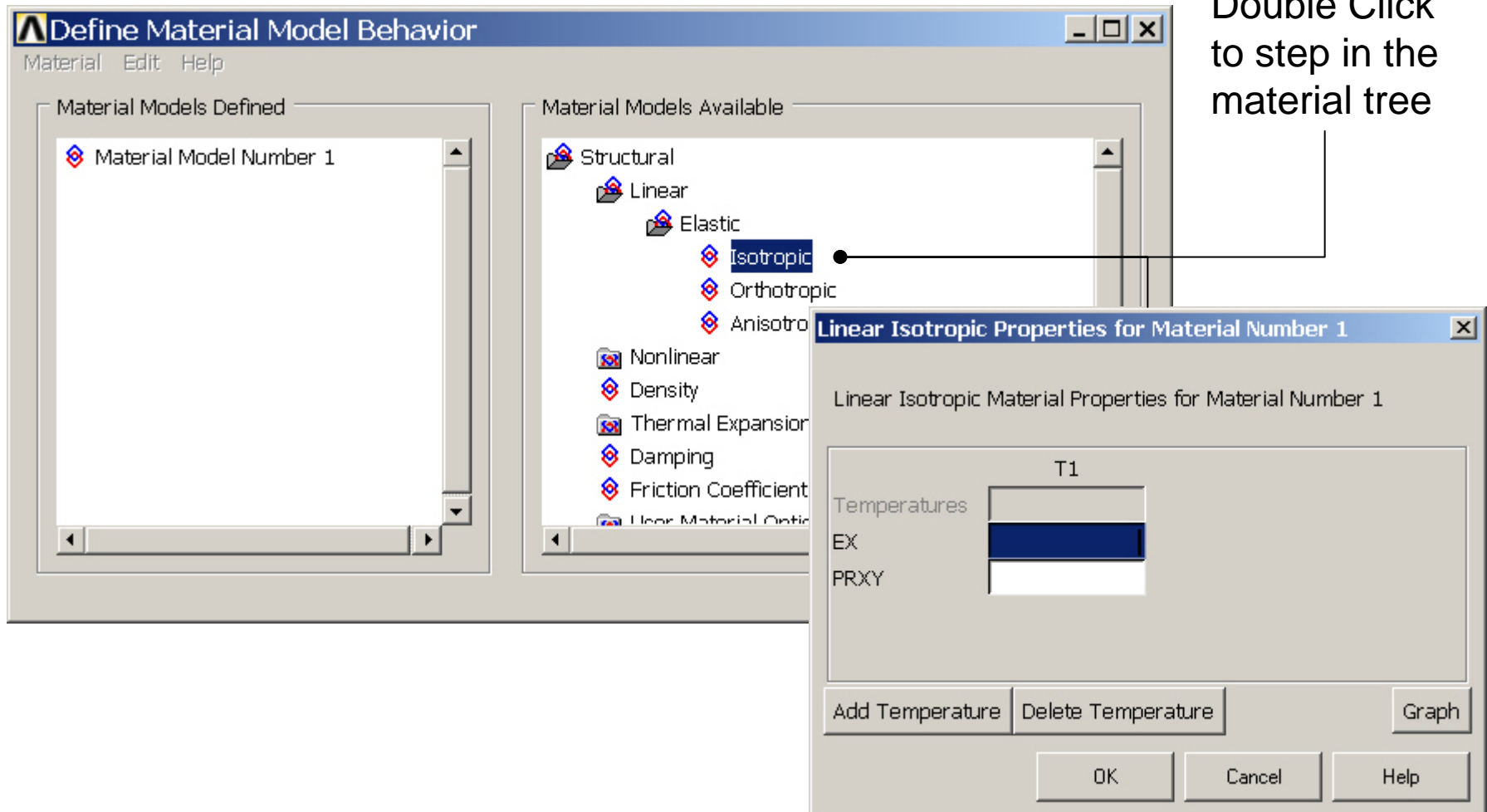
Select KP3

Press OK to finish



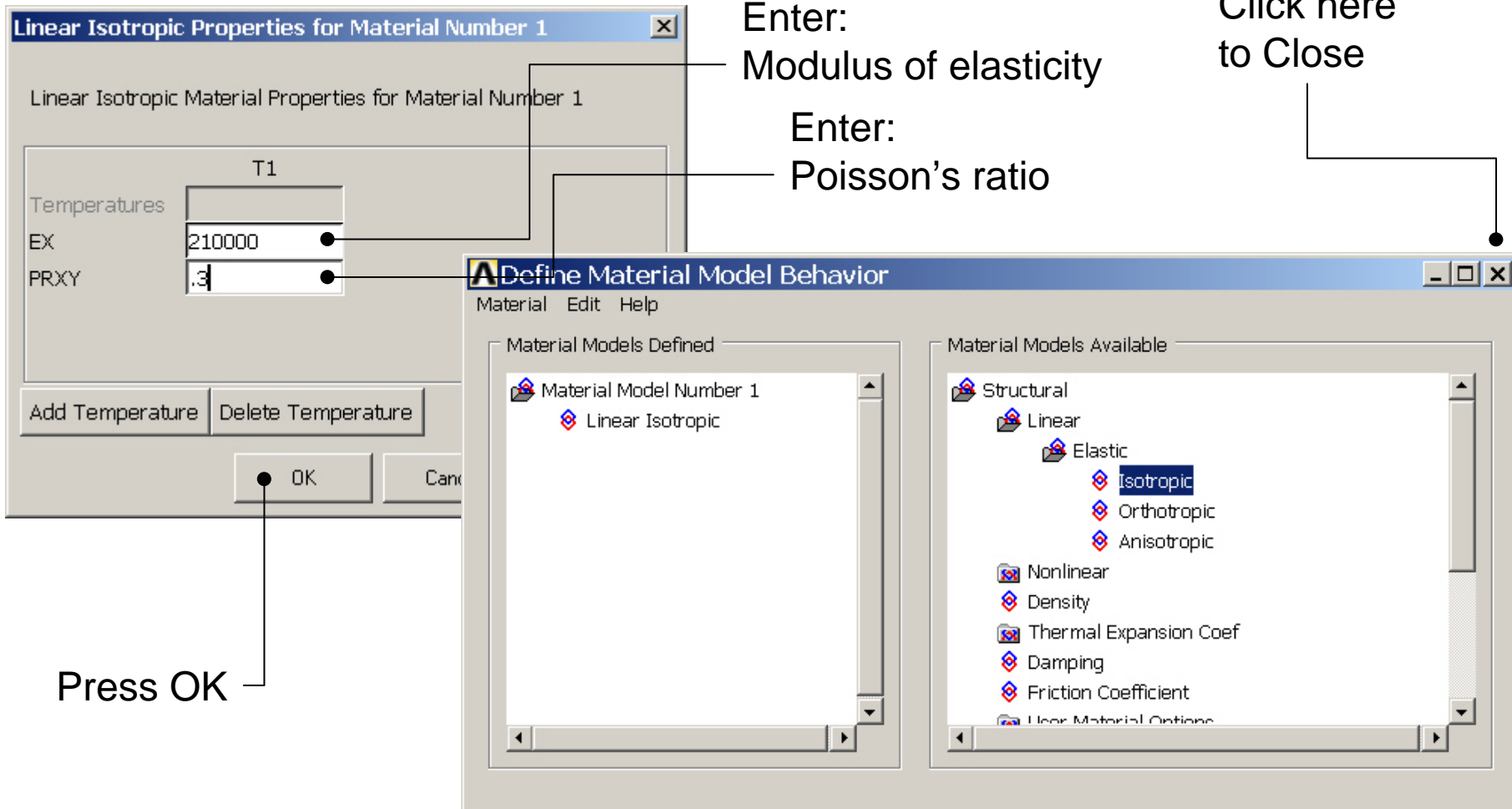
# Example - Material Properties

Preprocessor > Material Props > Material Models



# Example - Material Properties

Preprocessor > Material Props > Material Models



# 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

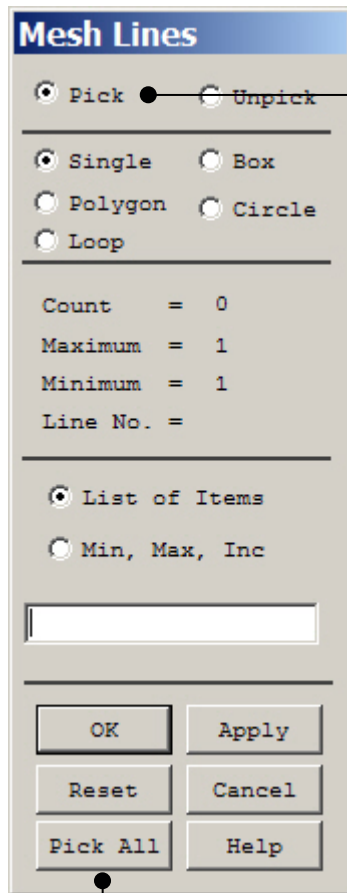
OK Apply Cancel Help

Press OK when finish with selection

Enter 5

# Example - Meshing

Preprocessor > Meshing > Mesh > Lines

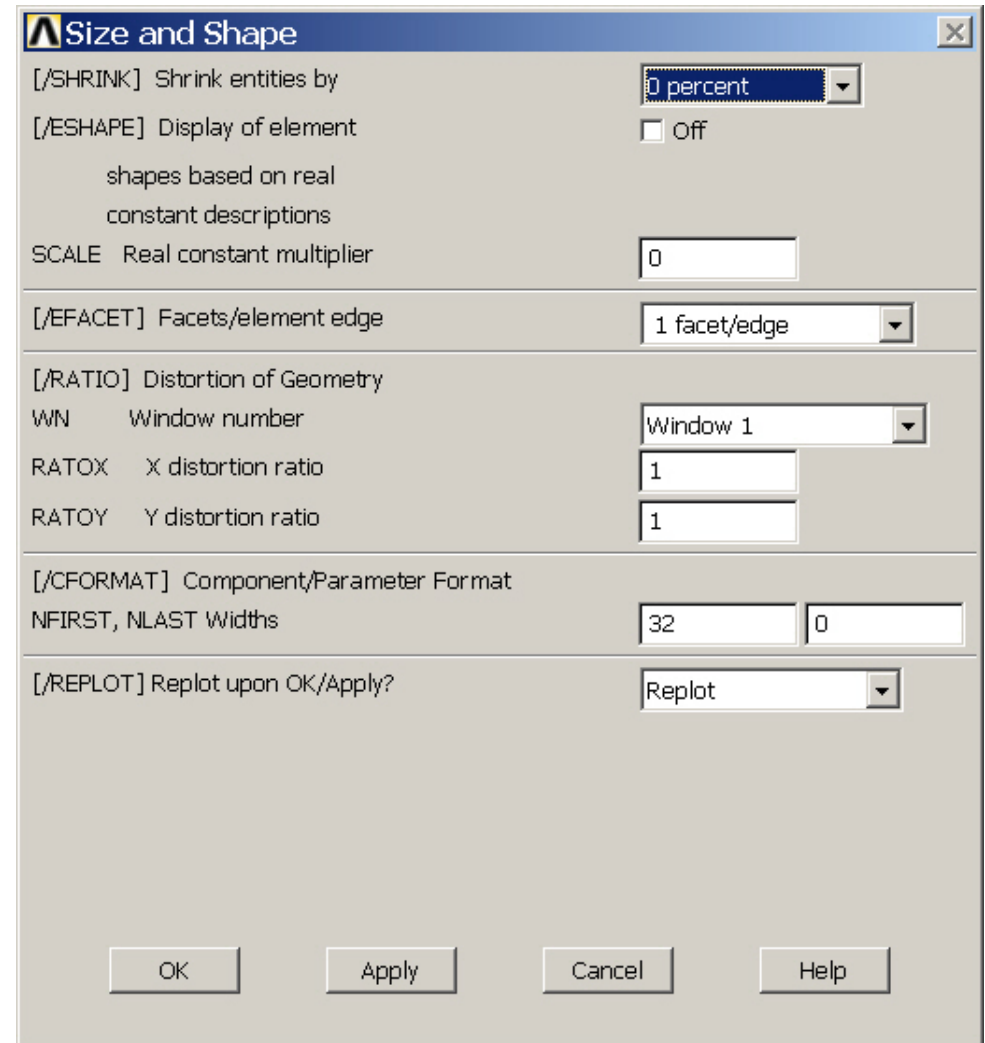
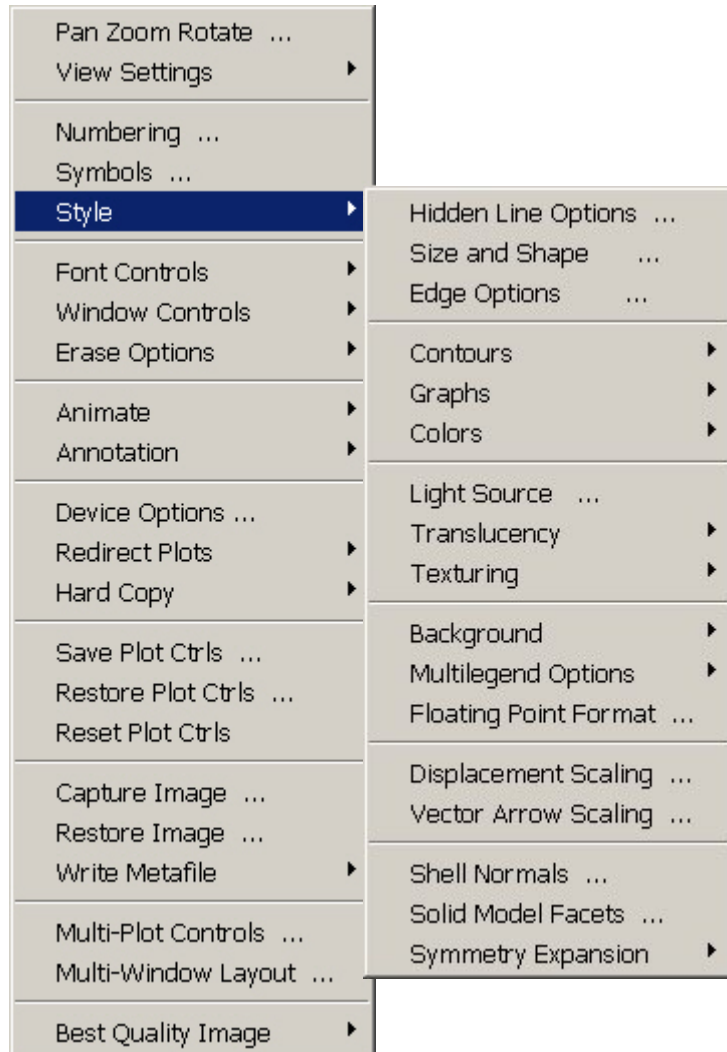


Select individual lines to be meshed by Picking

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

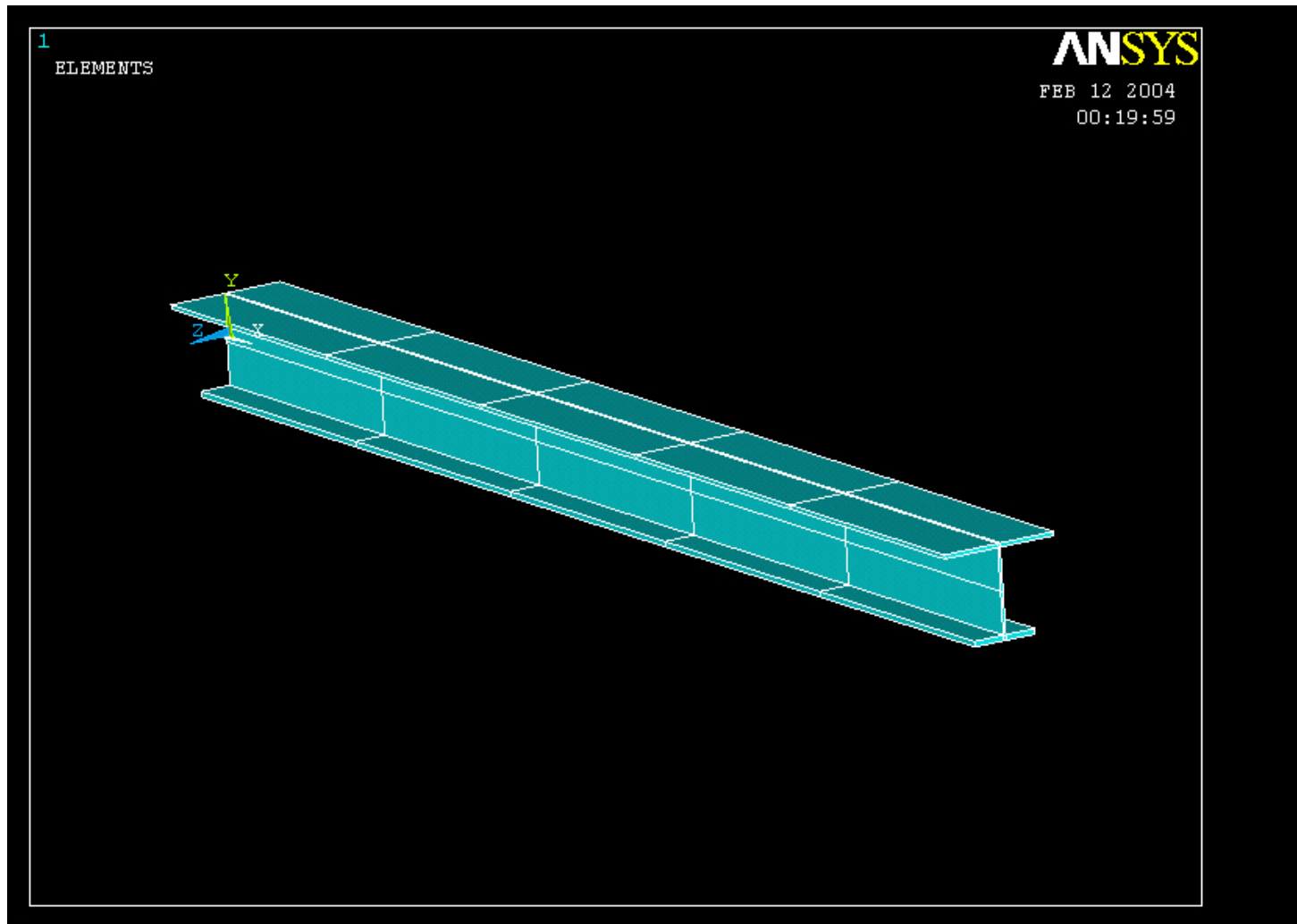
Select all lines defined to be meshed

# Example - PlotCtrls Menu



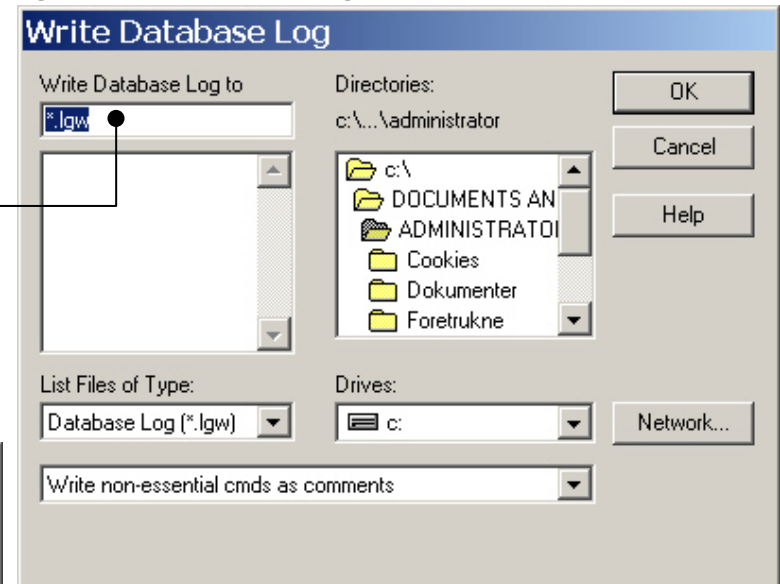


# Example – Display of Element

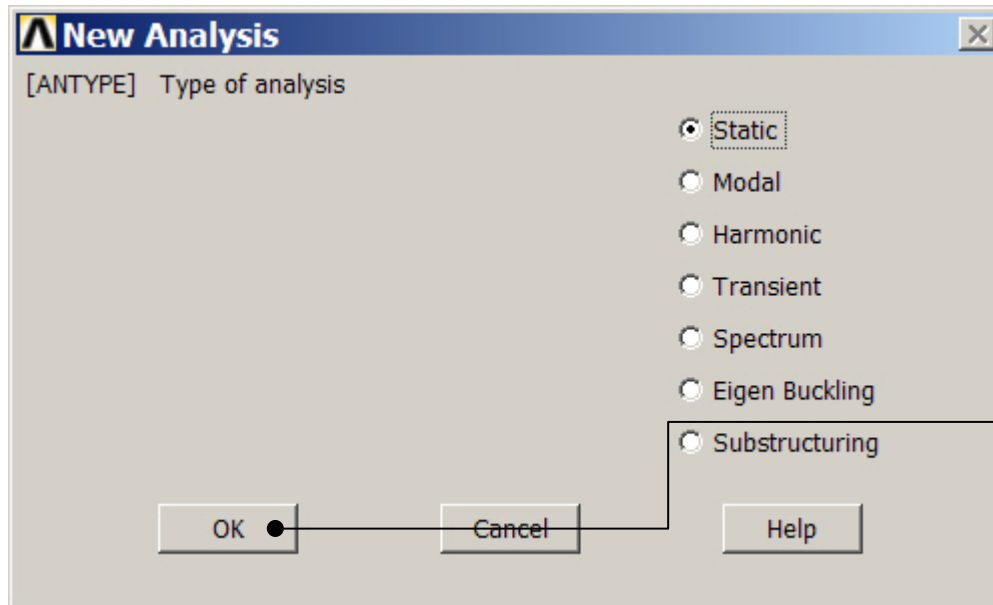


# Example – Analysis Type

**File > Write DB log file**  
Enter “example0504.lgw”



**Solution > Analysis Type > New Analysis**



Press OK

# Static solution – Analysis Options

The image shows two windows from the ANSYS software. The left window is the 'ANSYS Main Menu' with a tree view of options. The right window is the 'Static or Steady-State Analysis' dialog box.

**ANSYS Main Menu (Left):**

- Preferences
- Preprocessor
- Solution
  - Analysis Type
    - New Analysis
    - Restart
    - Sol'n Controls
  - Define Loads
  - Load Step Opts
  - Solve
  - FSI Set Up
  - Unabridged Menu
- General Postproc
- TimeHist Postpro
- Topological Opt
- ROM Tool
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

**Static or Steady-State Analysis (Right):**

**Nonlinear Options**

- [NLGEOM] Large deform effects: ☐ Off
- [NROPT] Newton-Raphson option: Program chosen
- Adaptive descent: ON if necessary

**Linear Options**

- [LUMPM] Use lumped mass approx?: ☐ No
- [EQSLV] Equation solver: Program Chosen
- Tolerance/Level -
- valid for all except Frontal and Sparse Solvers
- Multiplier -
- valid only for Precondition CG
- [PRECISION] Single Precision - ☐ Off
- valid only for Precondition CG
- [MSAVE] Memory Save - ☐ Off
- valid only for Precondition CG
- [PIVCHECK] Pivots Check: ☒ On
- valid only for Frontal, Sparse and PCG Solvers
- [SSTIF][PSTRES] Stress stiffness or prestress: None
- Note: If NLGEOM,ON then set SSTIF,ON.
- [TOFFST] Temperature difference-
- between absolute zero and zero of active temp scale

**Select Prestress ON**

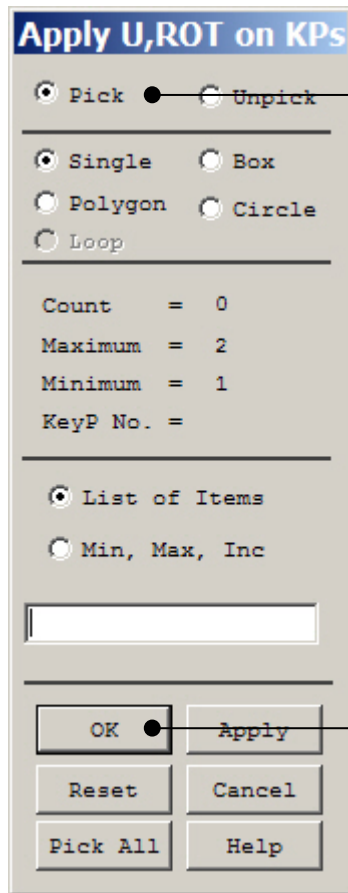
**Activate the Unabridged menu**

OK Cancel Help

Example0504

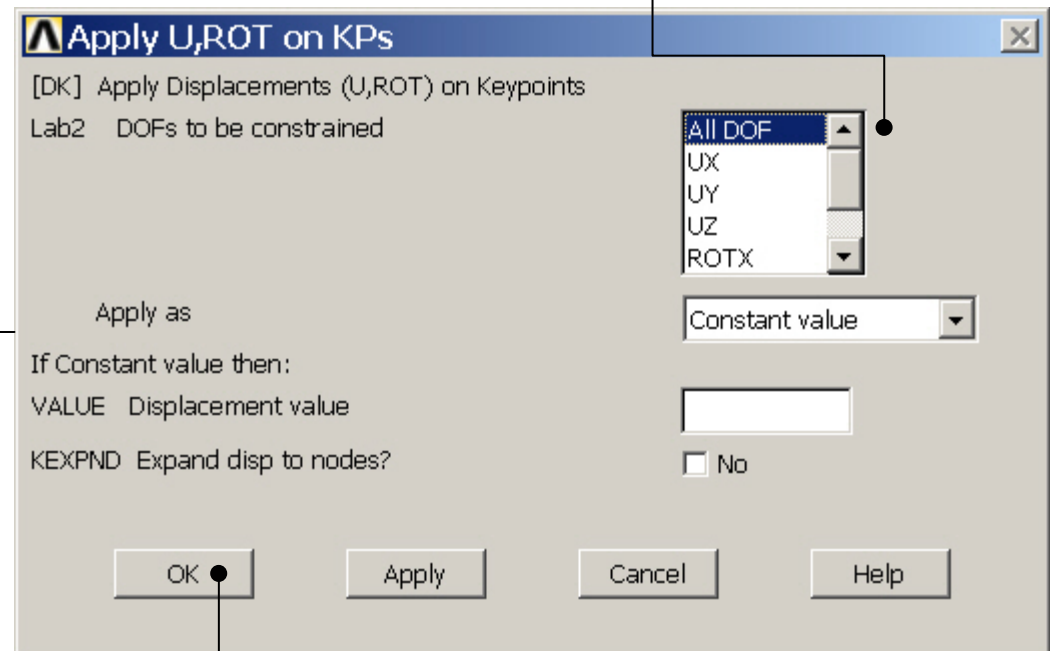
# Example – Define Loads

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



Select keypoint 1

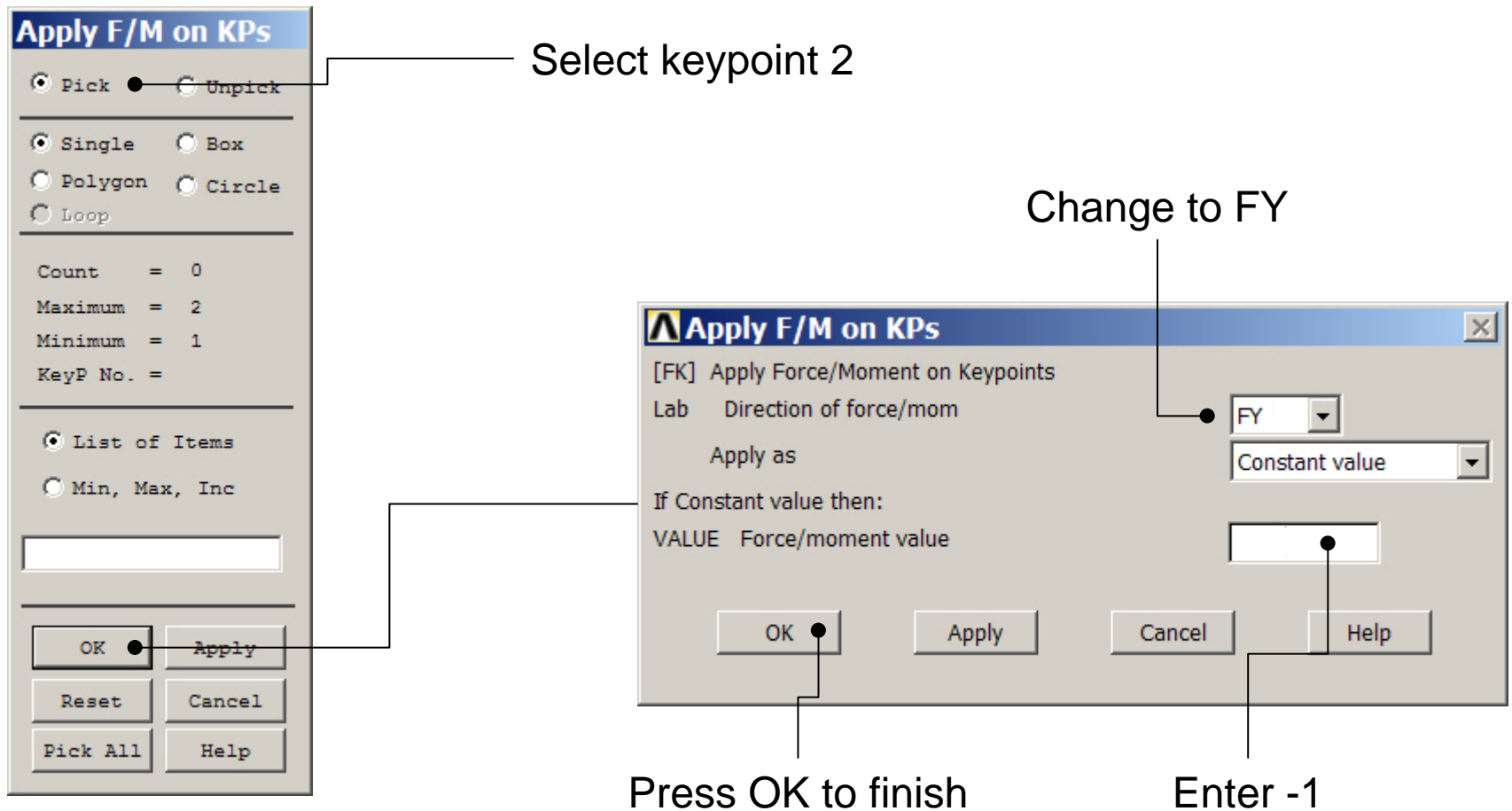
Select All DOF to fix/clamp the beam



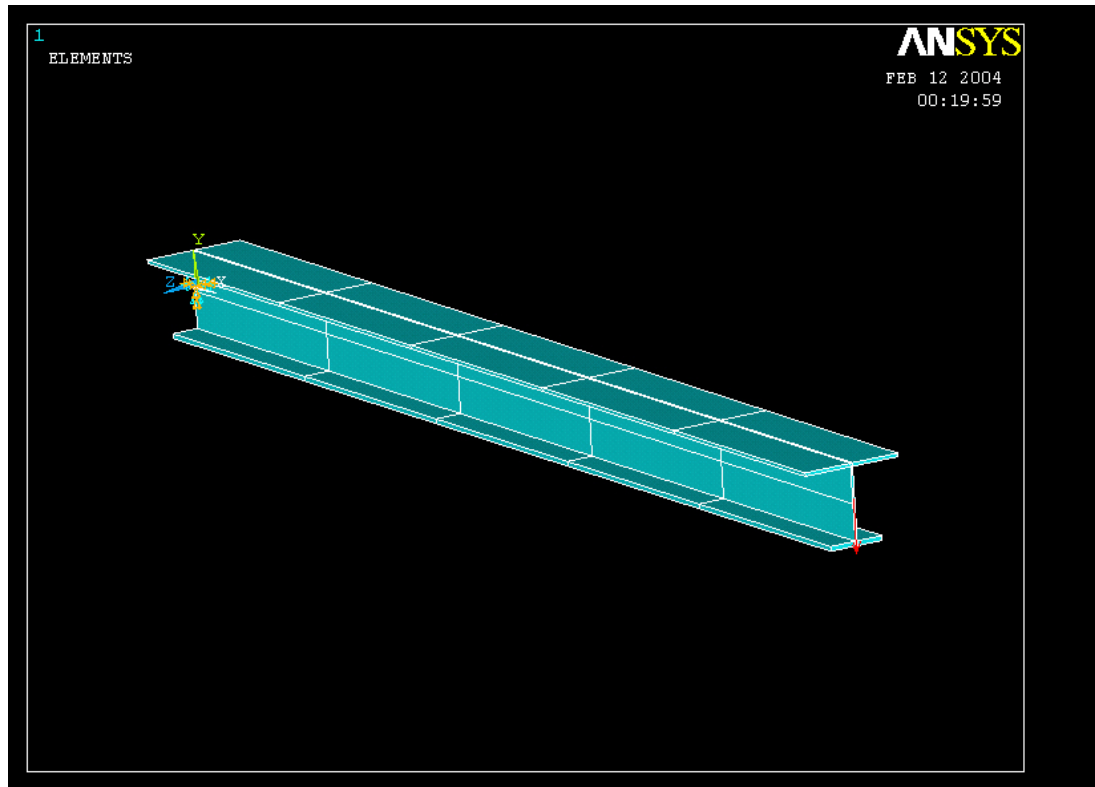
Press OK

# Example – Define Loads

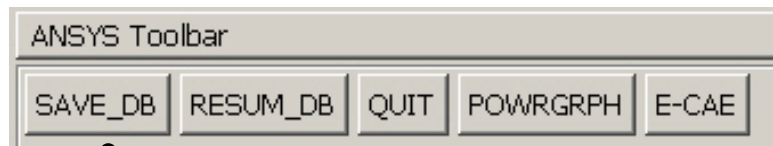
**Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints**



# Example - Save



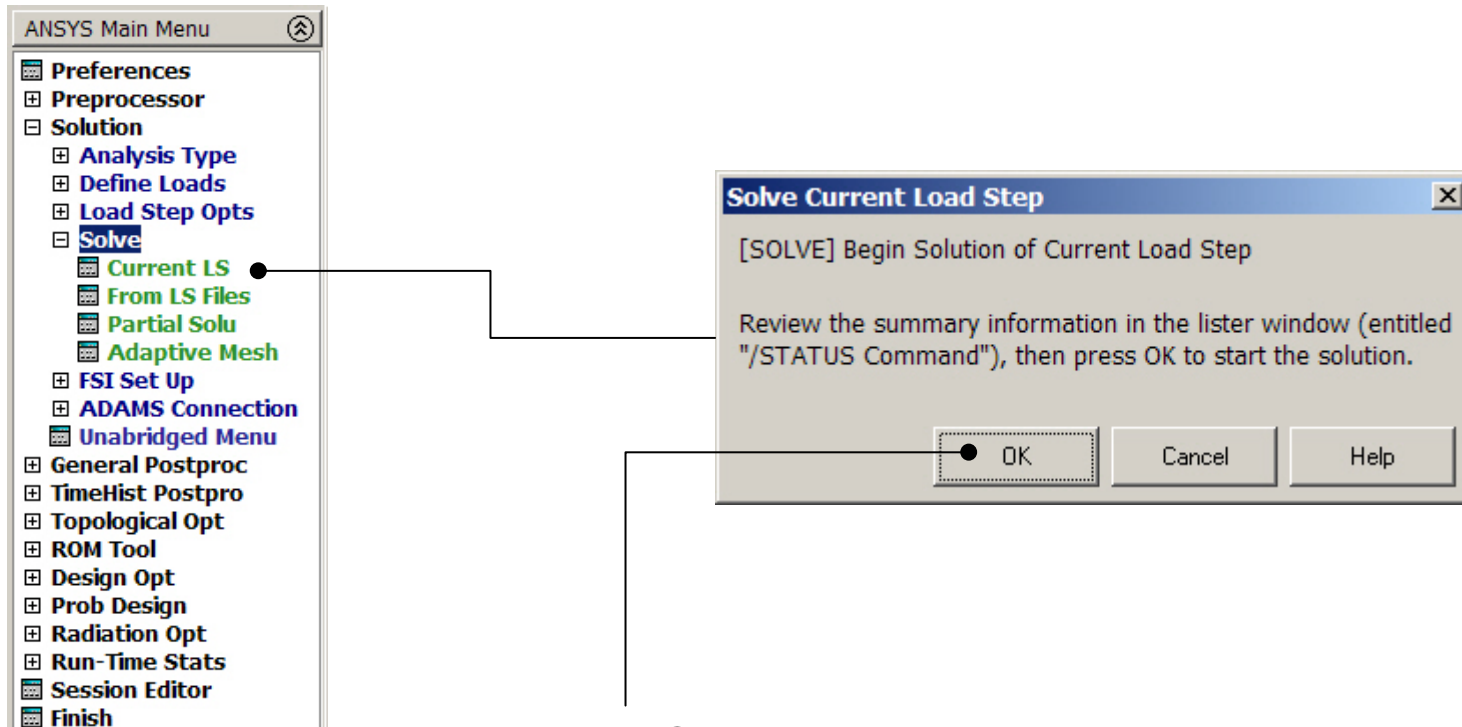
Display of Analysis model



Save the model

# Example - Solve

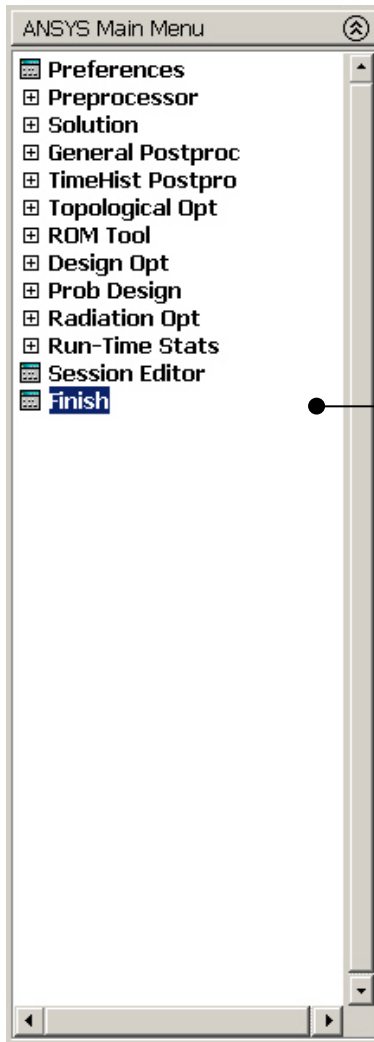
**Solution > Solve > Current LS**



Press OK

Example0504

# Example - Finish

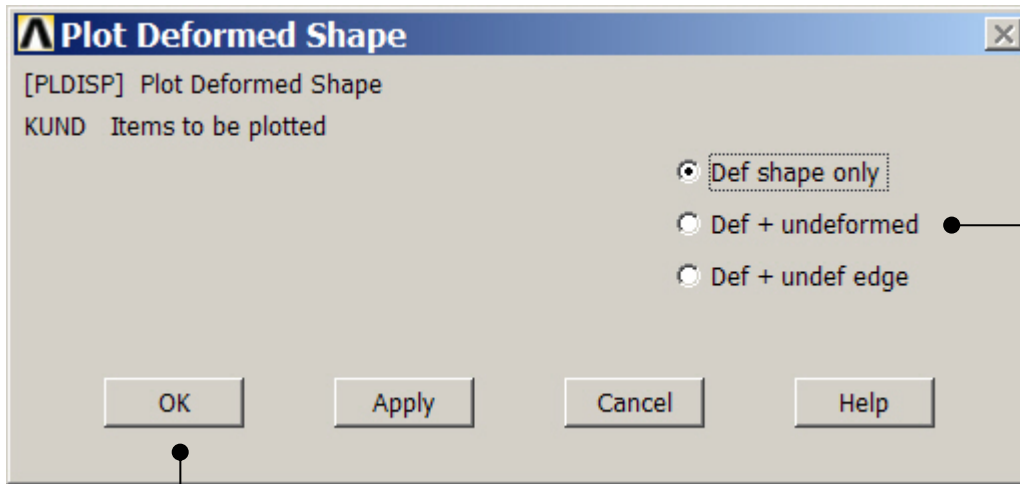


Press Finish to end the static solution



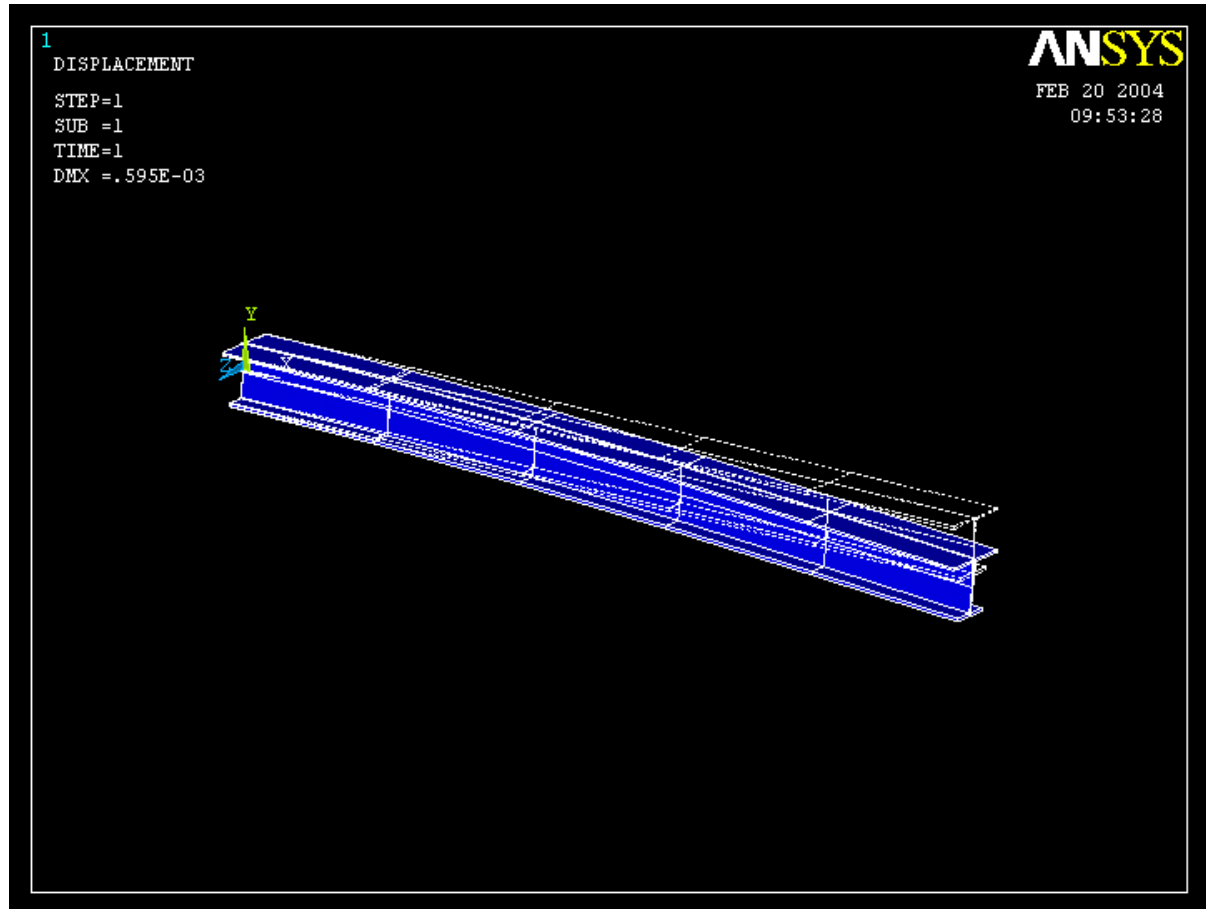
# Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Select "Def+undeformed"  
and Press OK

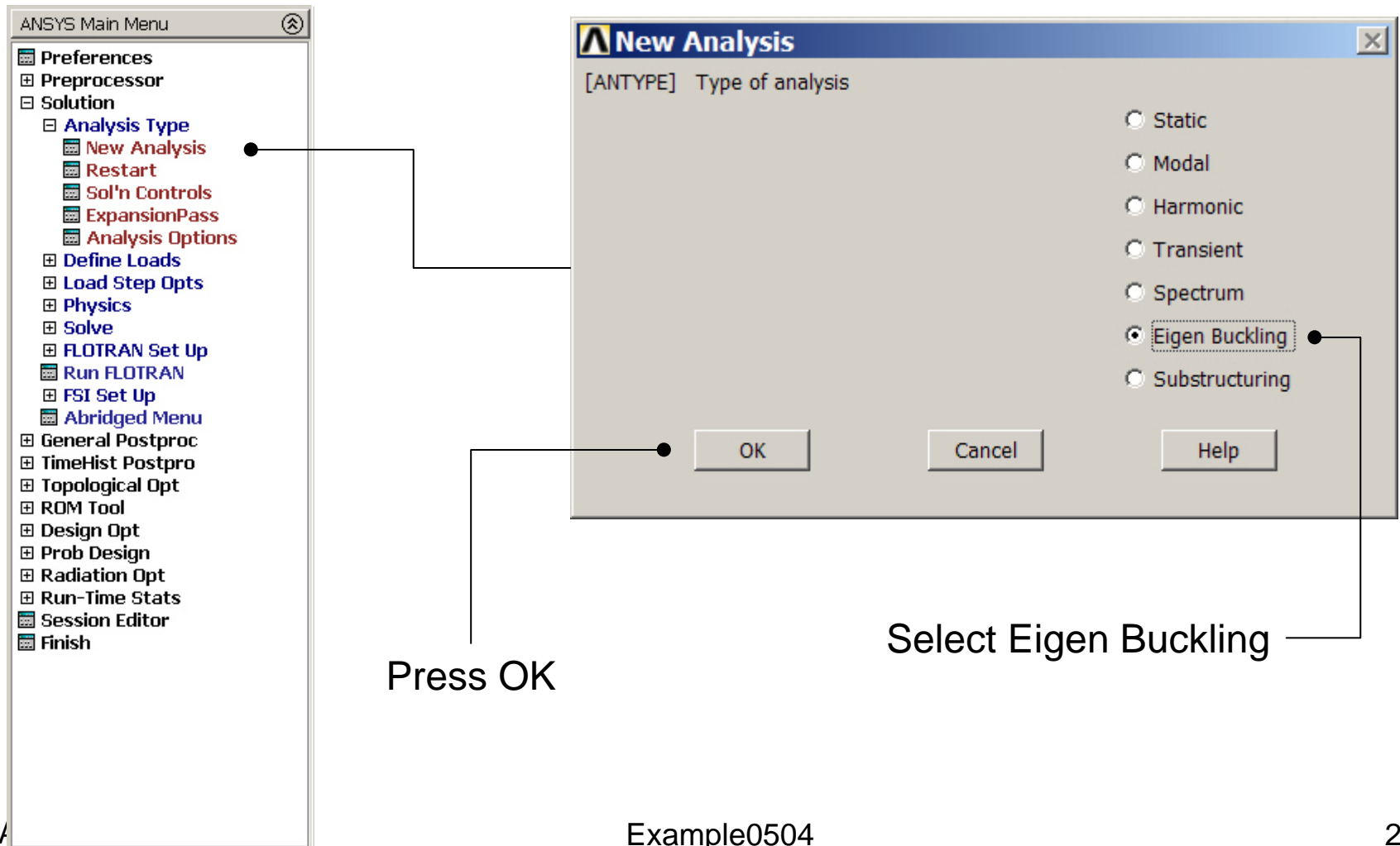
# Example - PostProcessing



Read Maximum displacement: DMX

# Eigen Buckling - New Analysis

Main Menu> Solution> Analysis Type> New Analysis

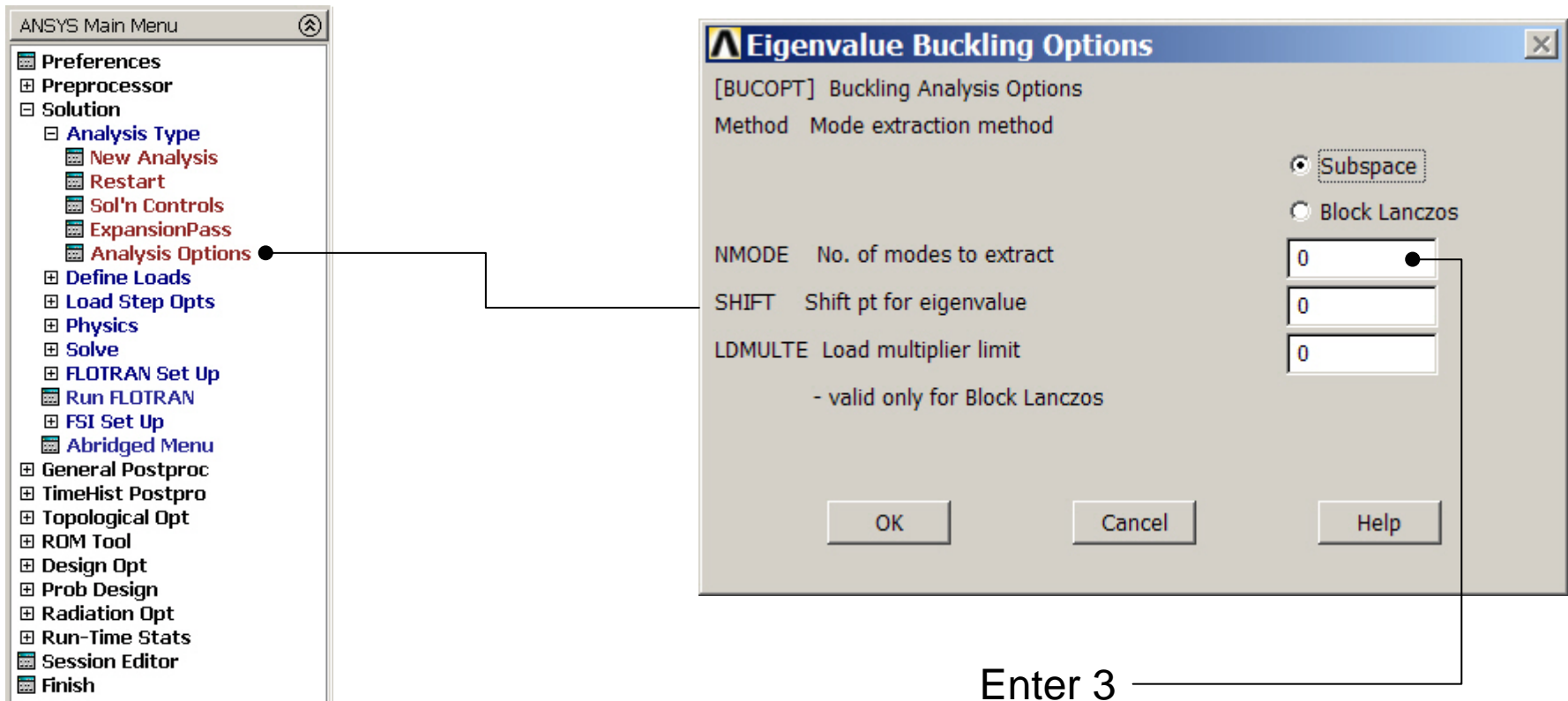


Example0504

27

# Eigen Buckling – Analysis Options

Main Menu> Solution> Analysis Type> Analysis Options



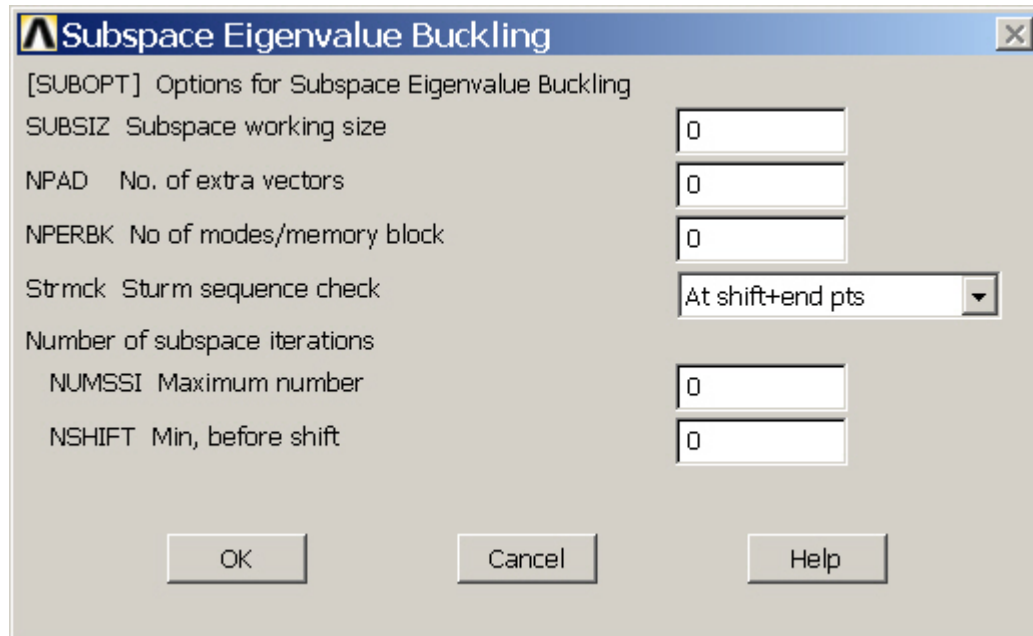
Example0504

28

# Example - Shifting

- In some cases it is desirable to shift the values of eigenvalues either up or down. These fall in two categories:
  - Shifting down, so that the solution of problems with rigid body modes does not require working with a singular matrix.
  - Shifting up, so that the bottom range of eigenvalues will not be computed, because they had effectively been converted to negative eigenvalues. This will, in general, result in better accuracy for the higher modes.

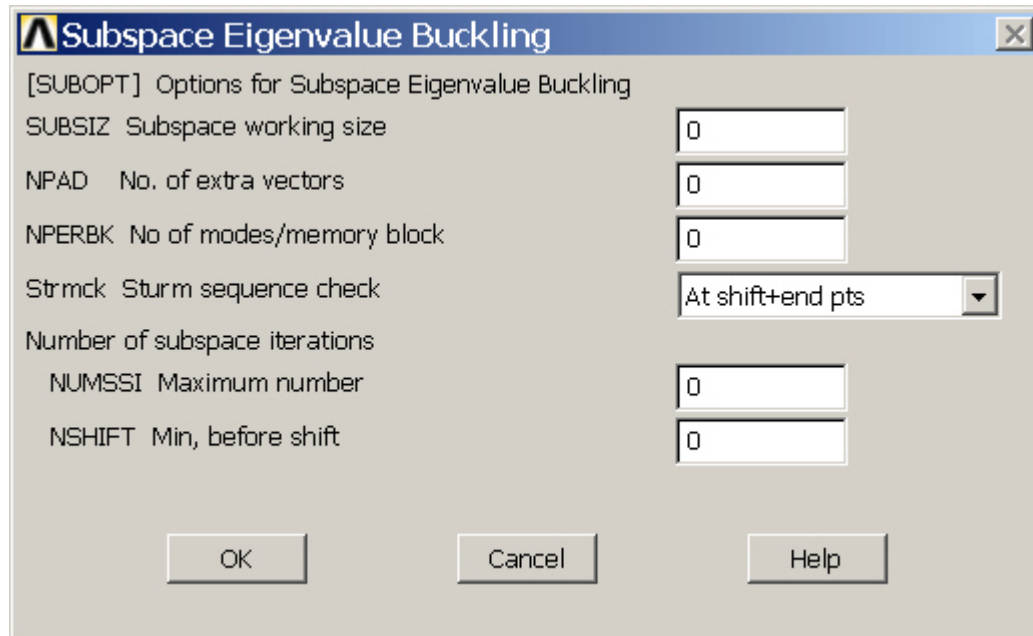
# Example – Subspace Options



Subspace working size. Defaults to  $NMODE + 4$  (where  $NMODE$  is input on the [MODOPT](#) or [BUCOPT](#) command). Minimum is 8. Maximum is  $NMODE + NPAD$ . The larger the value, the smaller the number of iterations (but more time per iteration).

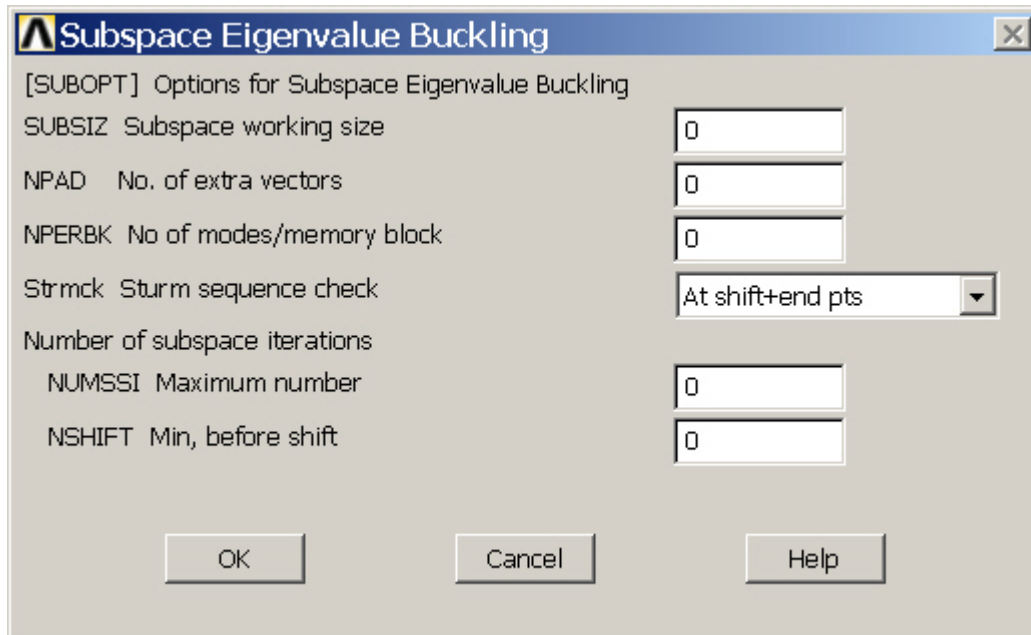
Number of extra vectors used in the iterations. Defaults to 4. The total number of vectors used is  $NMODE + NPAD$ .

# Example – Subspace Options



Number of modes per memory block. If 0 (or blank), perform data management in-memory for all modes (no disk I/O). If greater than zero, use some disk I/O (slower for decreasing *NPERBK* values, but may be needed for large problems). The minimum nonzero value is the number of degrees of freedom per node for the model.

# Example – Subspace Options

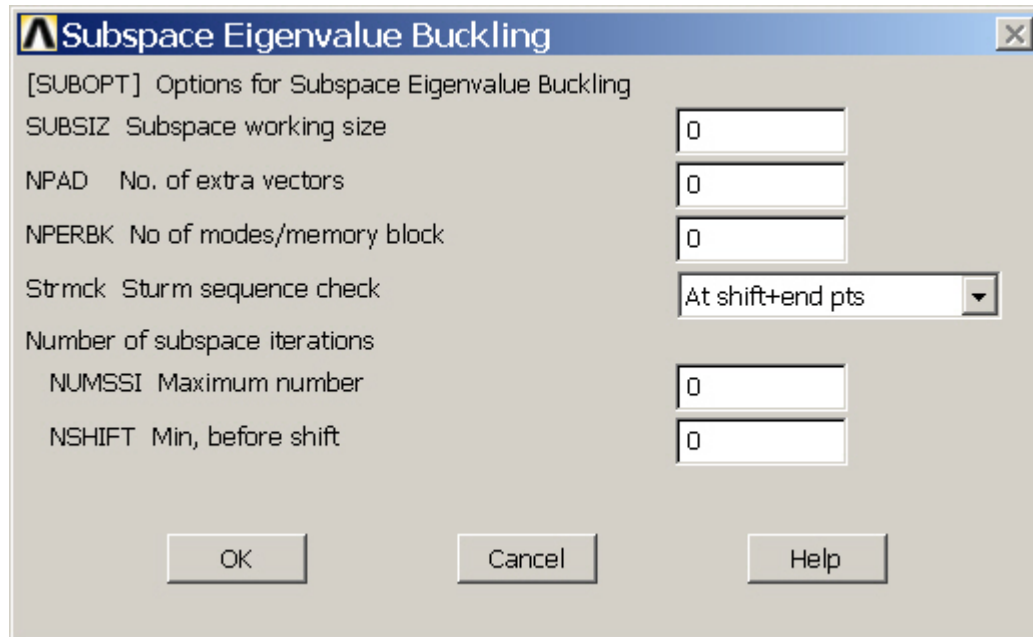


Maximum number of subspace iterations (defaults to 100). Fewer iterations will be done if convergence occurs before the 100th iteration. Convergence occurs whenever the normalized change in the eigenvalue calculations between successive iterations for the first *NMODE* eigenvalues is less than 1.0E-5.

Minimum number of subspace iterations completed before a shift is performed. The default is 5 and the minimum is 2. Use *FREQB* on the [\*\*MODOPT\*\*](#) command or *SHIFT* on the [\*\*BUCOPT\*\*](#) command to define the initial shift point.



# Example – Subspace Options



Number of Jacobi iterations used per subspace iteration (used only with the JCG and PCG options on the [EQSLV](#) command). Defaults to the number of degrees of freedom divided by the maximum wave front for the model. The minimum is 5.

Sturm sequence check key:

ALL --

Perform check at all shift points as well as at the end point (default).

PART --

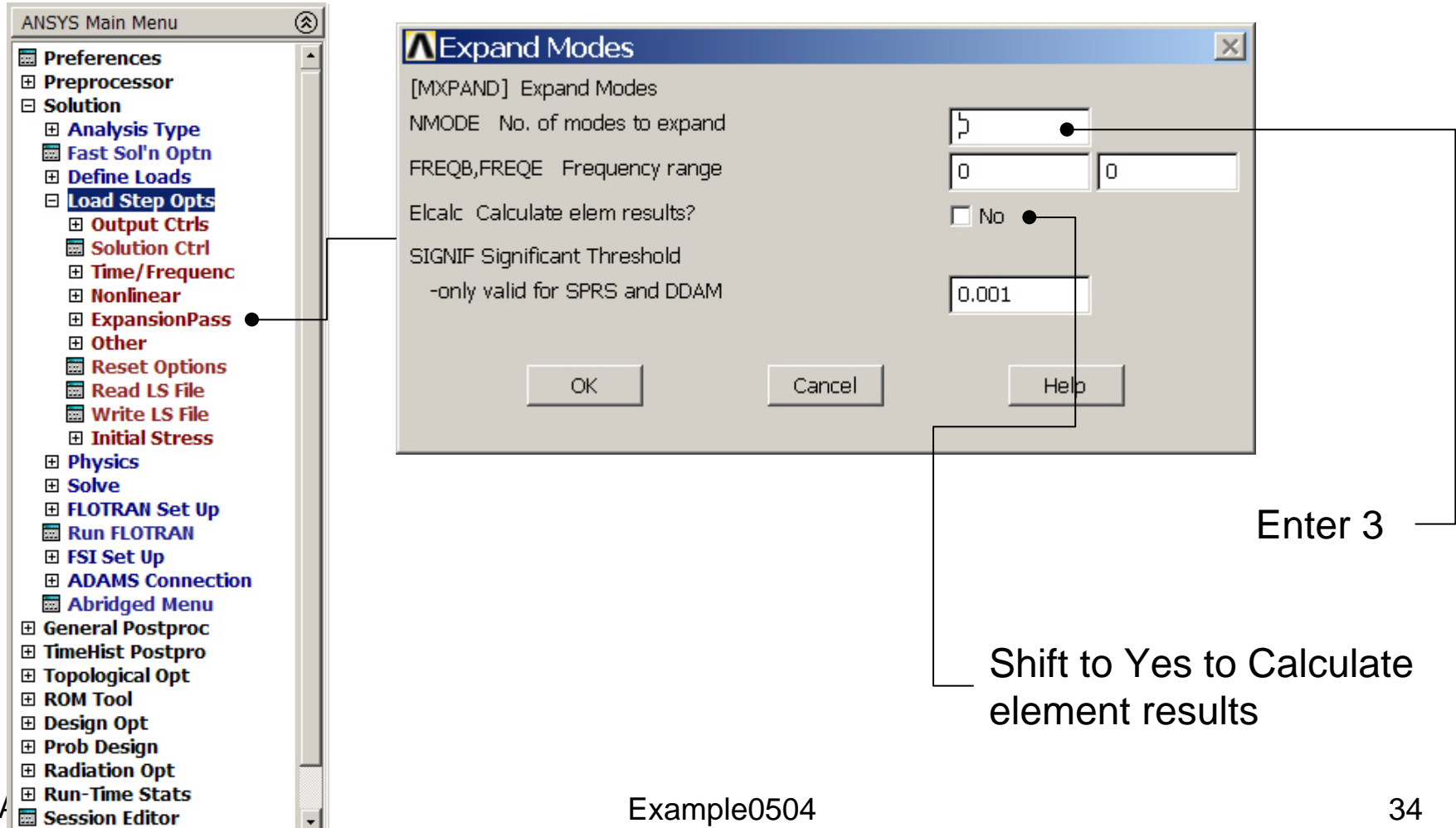
Perform check only at all shift points.

NONE --

Do not perform Sturm sequence check.

# Eigen Buckling – Expanding Modes

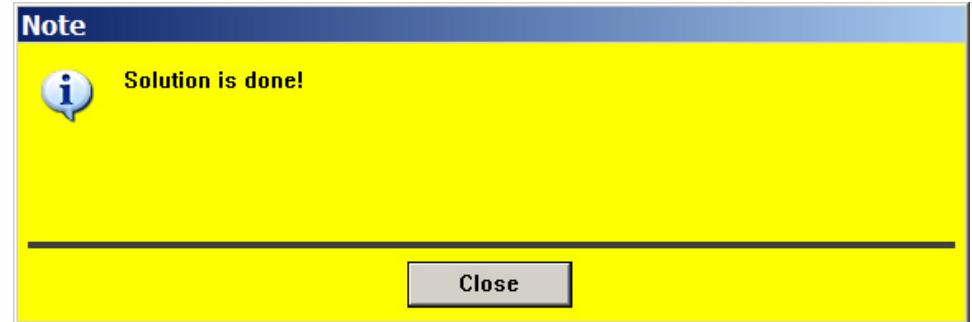
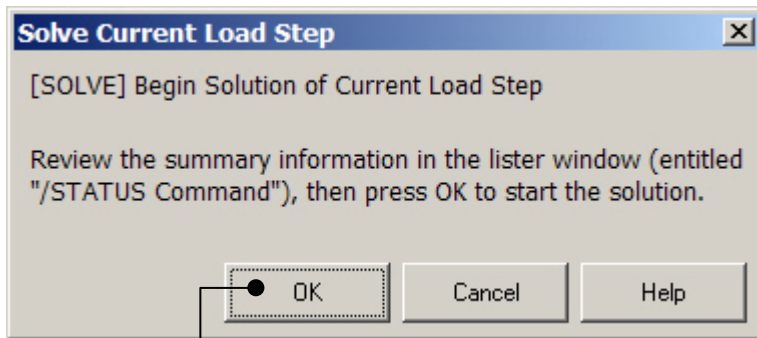
Main Menu> Solution> Load Step Opts > ExpansionPass >  
Single Expand > Expand Modes



Example0504

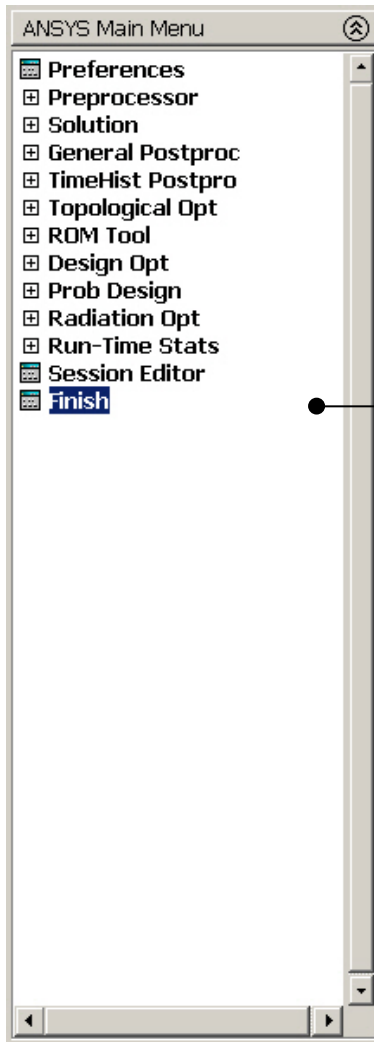
# Example - Solve

**Solution > Solve > Current LS**



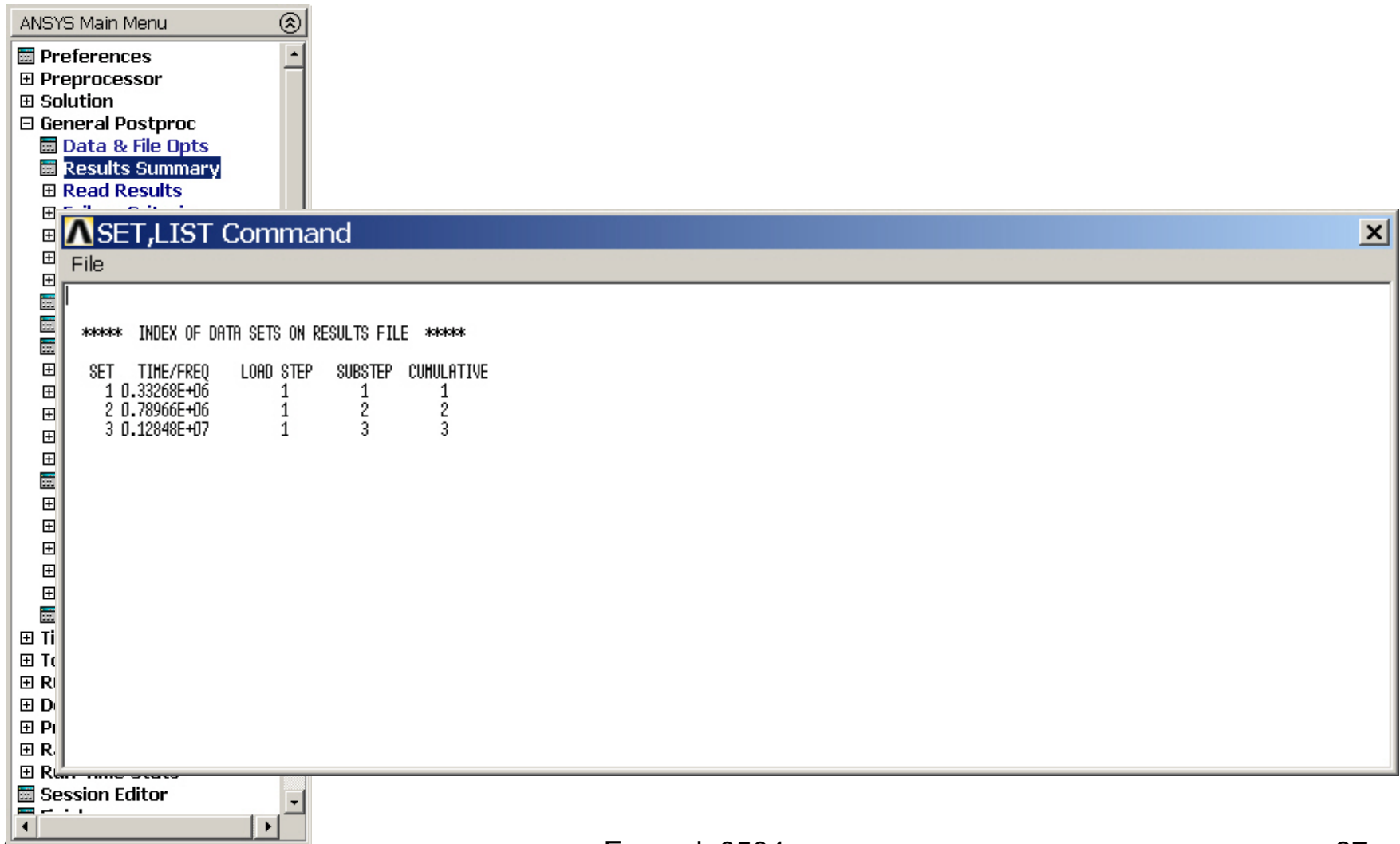
Press OK

# Example - Finish



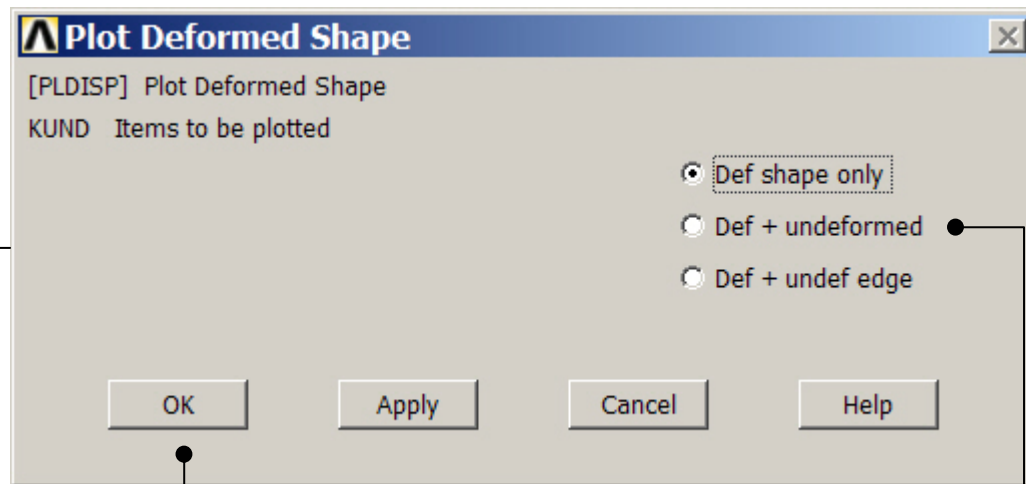
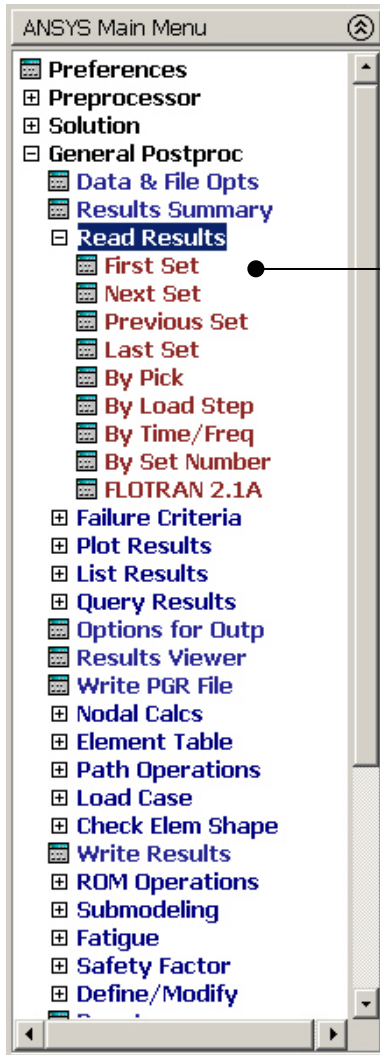
Press Finish to end the eigen buckling solution

# Example – Results Summary



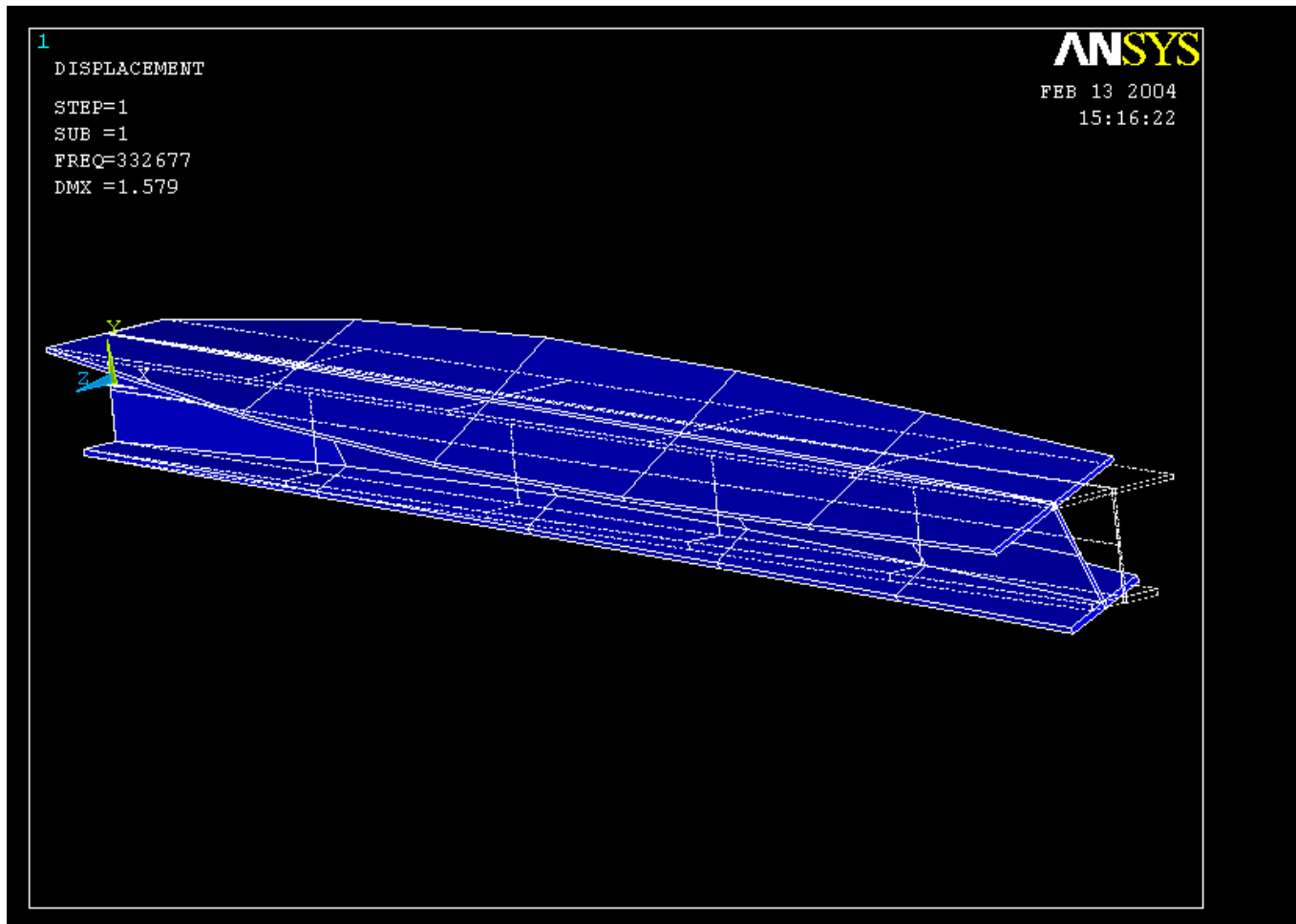
# Example – Read Results

General Postproc > Plot Results > Deformed Shape

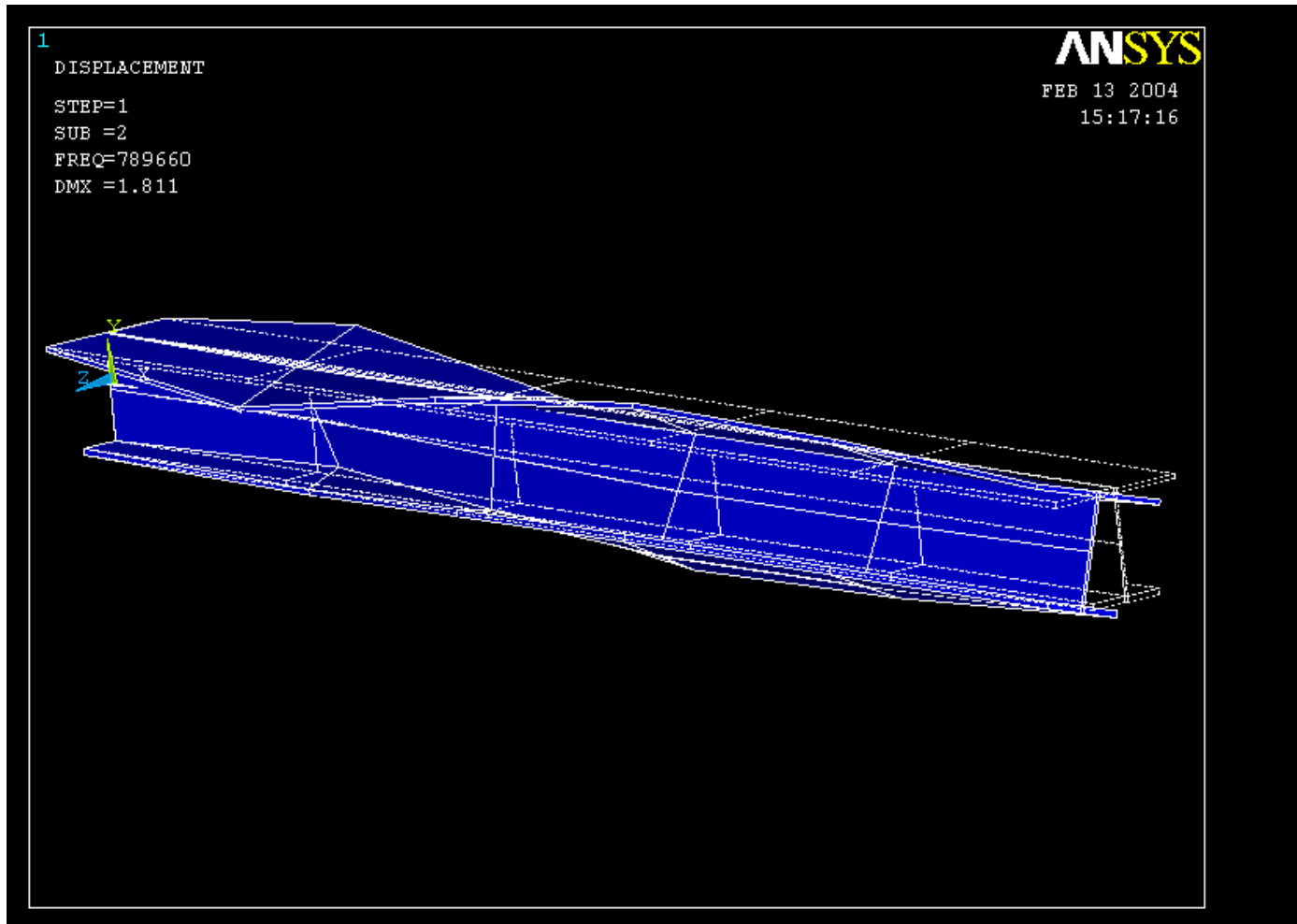


Select "Def+undeformed"  
and Press OK

# Example – First set



# Example – Next set





# Example – Next set

