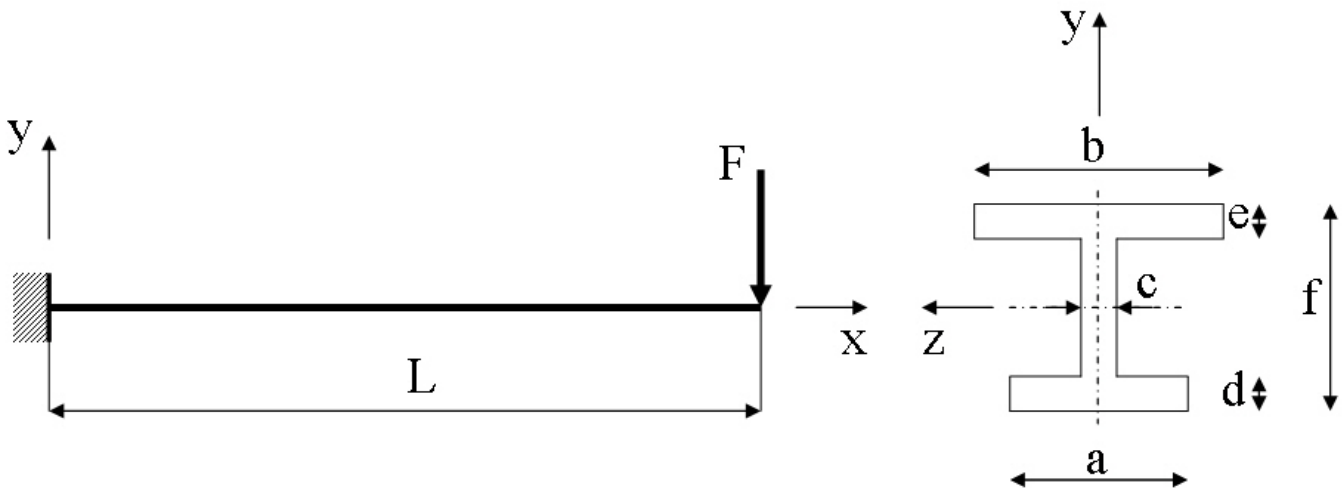
# Course in ANSYS

## Example0504

# Example – Cantilever beam



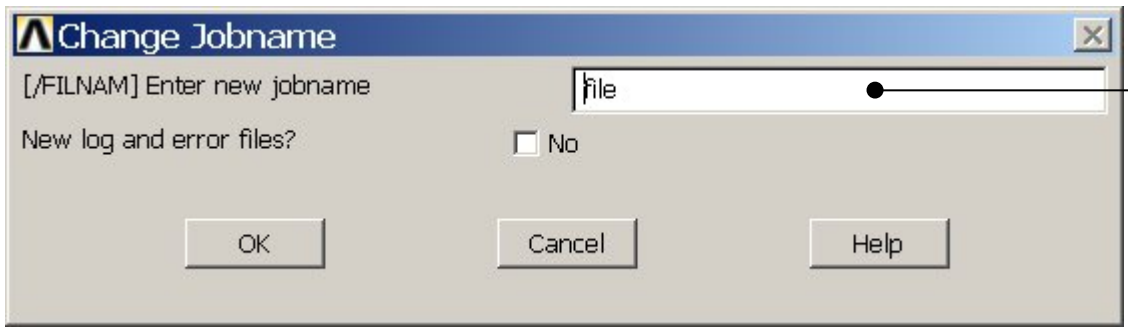

**Objective:**
Compute the buckling load
**Tasks:**
Display the deflection figure?
**Topics:**
Topics: Start of analysis, Element type, Real constants, Material, modeling, element size for beam models, saving/restoring, orientation keypoints

$E = 210000 N/mm^2$
$\nu = 0.3$
L = 5000mm
a = 250mm
b = 450mm
c = 10mm
d = 20mm
e = 15mm
f = 350mm
F = ?

# Example - title

**Utility Menu > File > Change Jobname** — GUI

/jobname, Example0504 — Command line entry

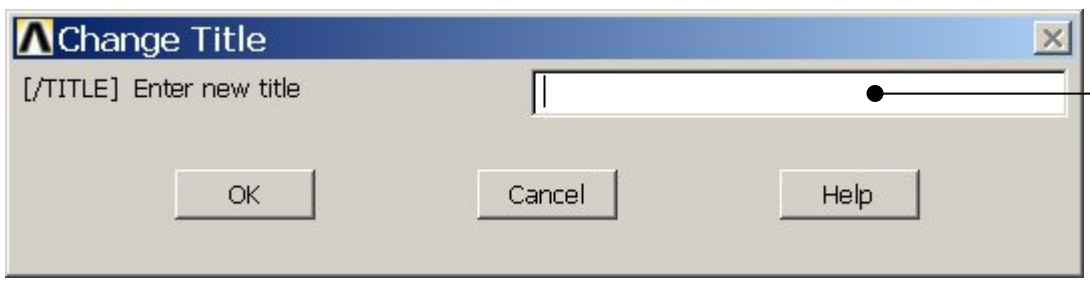



Enter: Example0504

**Utility Menu > File > Change Title**

/title, Cantilever beam

Enter: Cantilever beam



# Example - Keypoints

Note: An empty # result in automatic numbering.

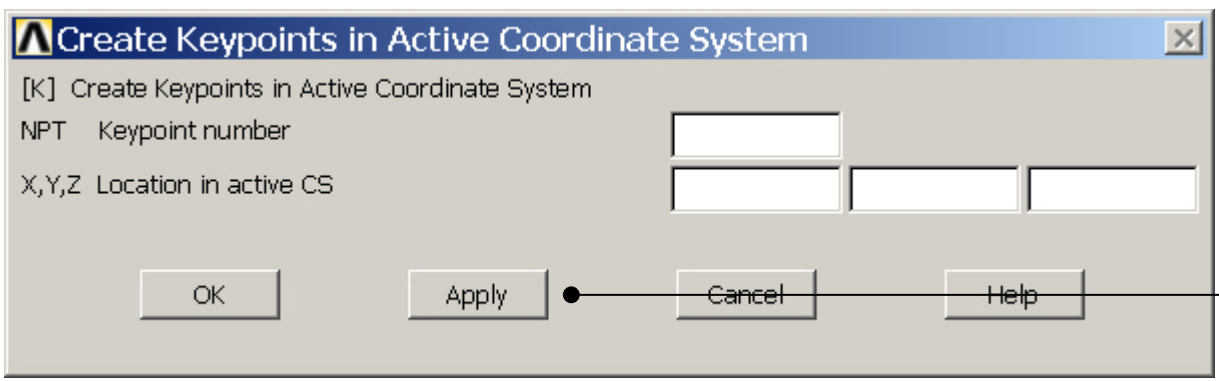
**Preprocessor > Modeling > Create > Keypoints > In Active CS**

/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**

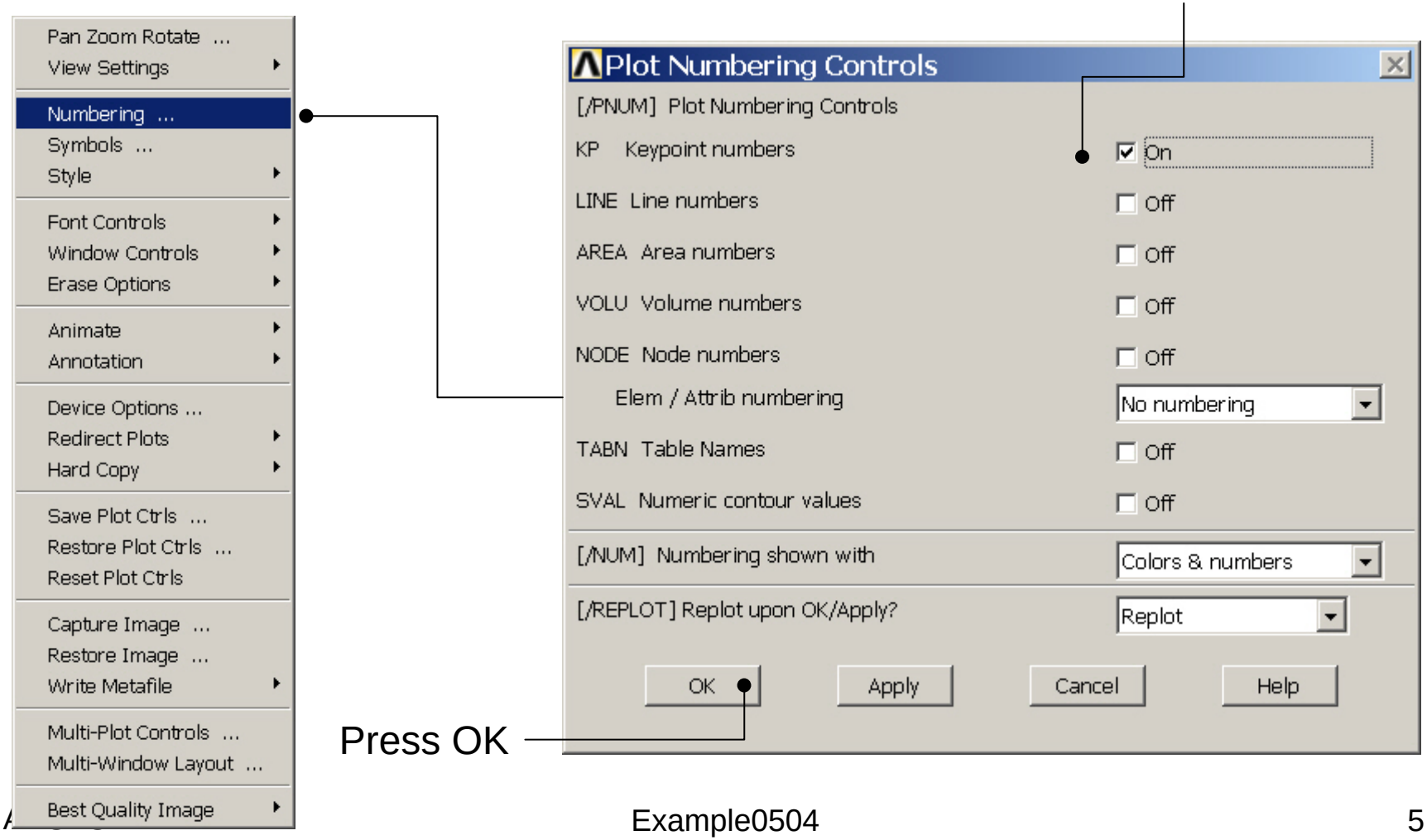



Note: An empty box result in a zero. It is allowed to enter 0.0 in each box.

# Example - Numbering

**Utility Menu > PlotCtrls > Numbering**

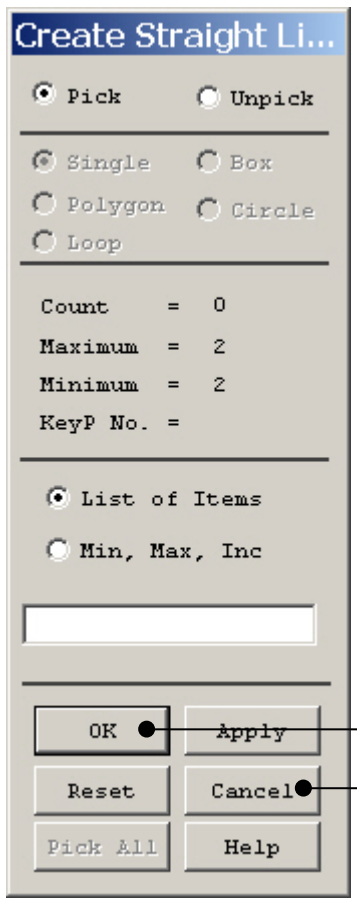
Switch on Keypoint numbers




Press OK

# 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:

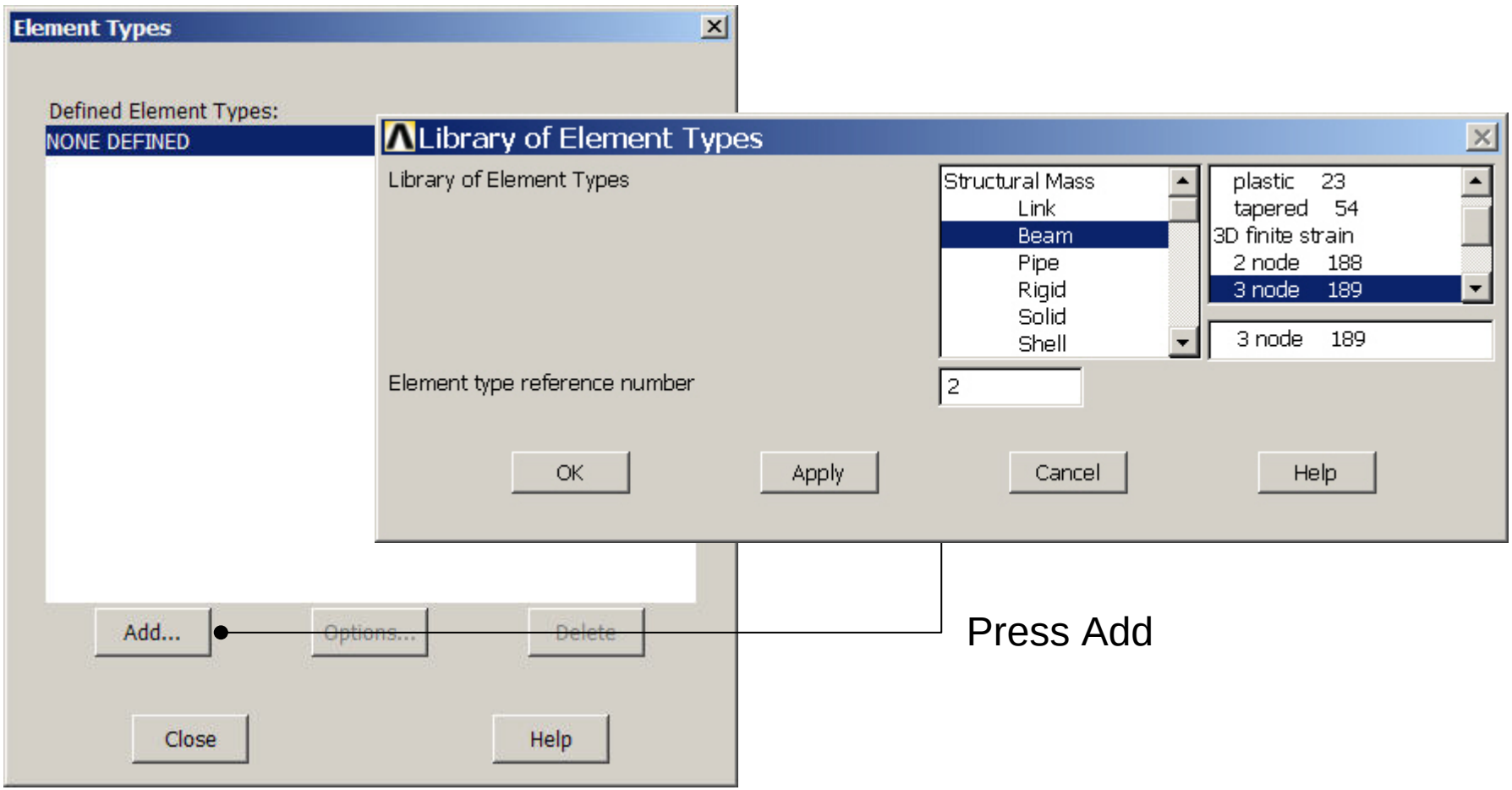
Pick: upward arrow

Unpick: downward arrow

Press OK or Cancel to finish selection

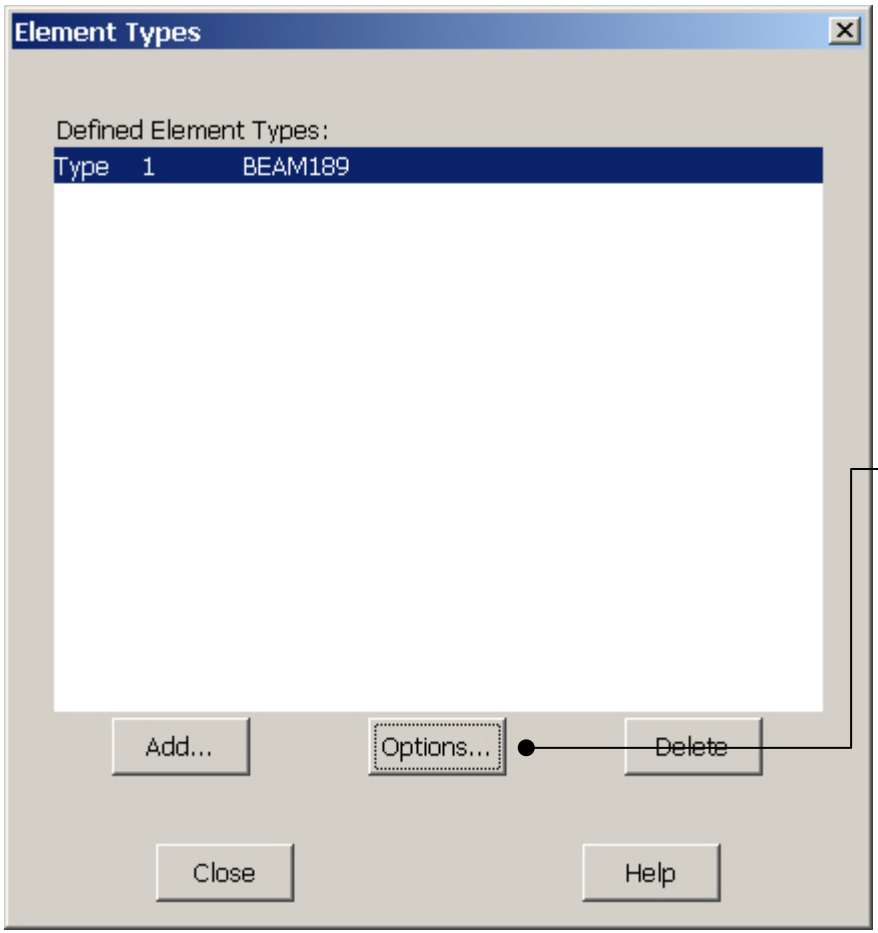
# Example – Element Type

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



# Example - Element Type

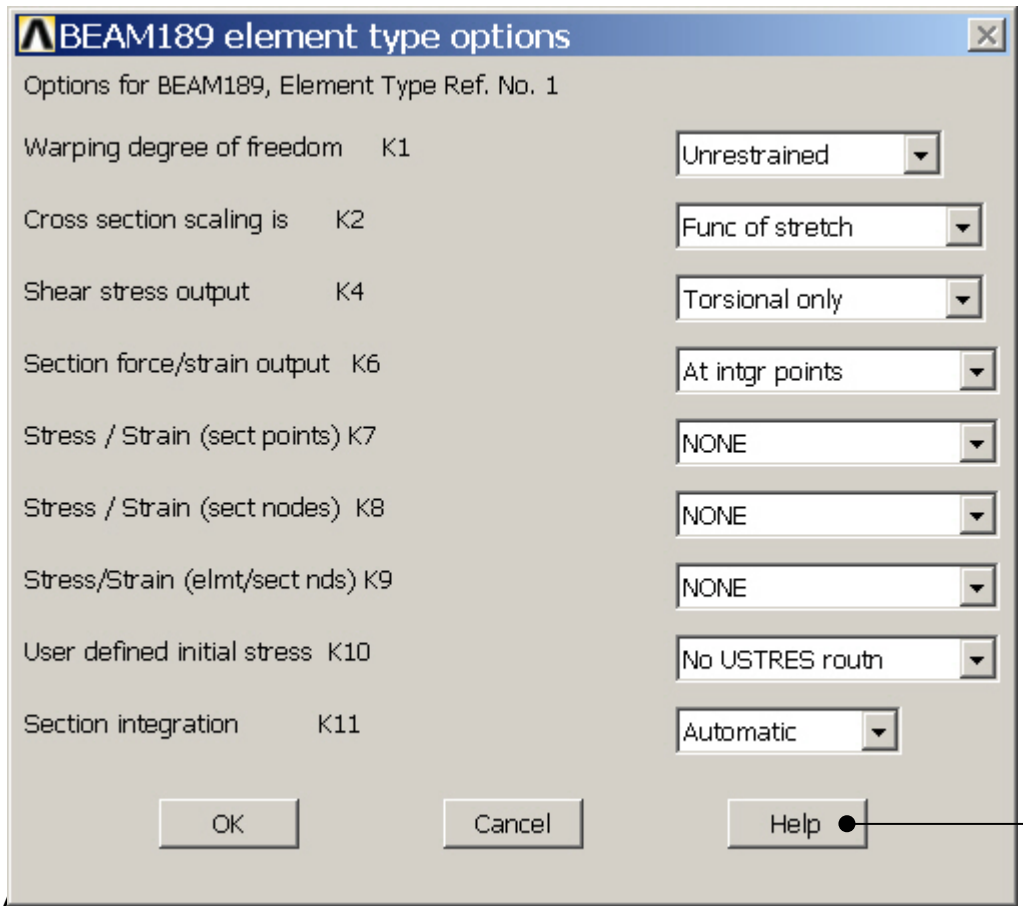
**Preprocessor > Element Type > Add/Edit/Delete**




Press Options

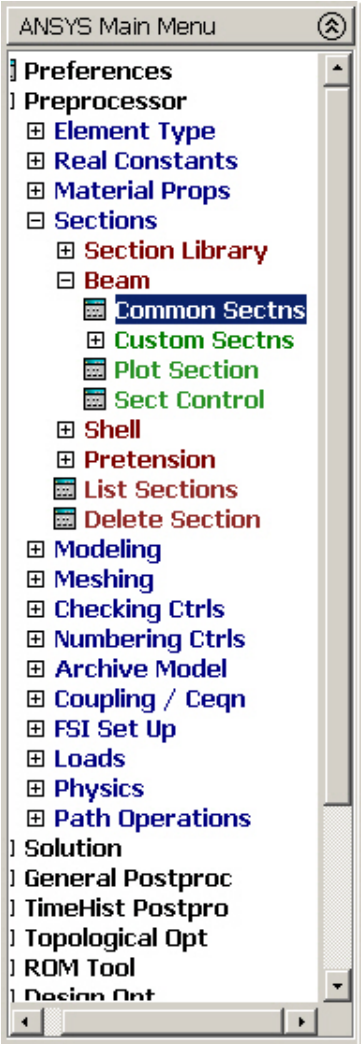
# Example - Element Type

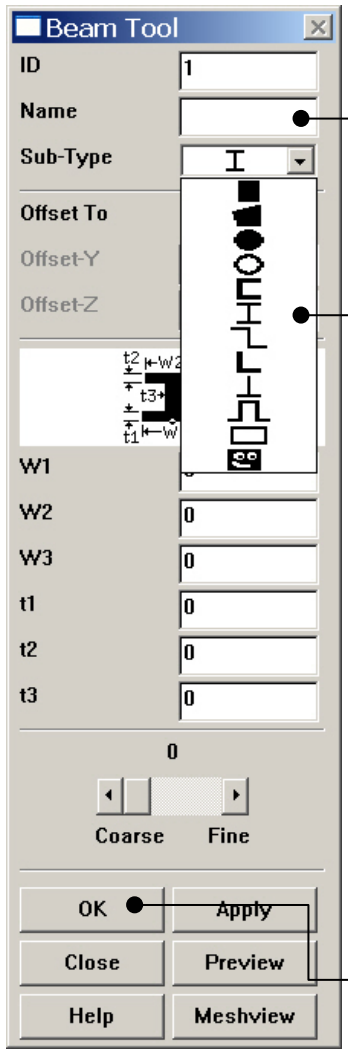
**Preprocessor > Element Type > Add/Edit/Delete**




Press Help to learn more about the element.

# Example - Section



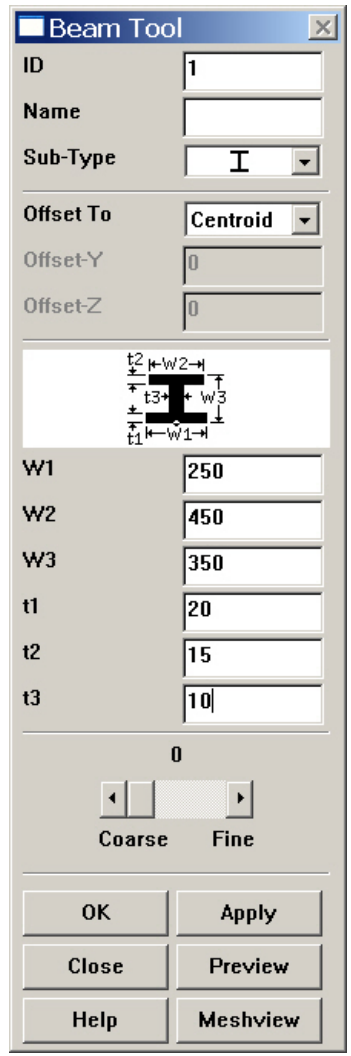




Enter i253

Select the I profile

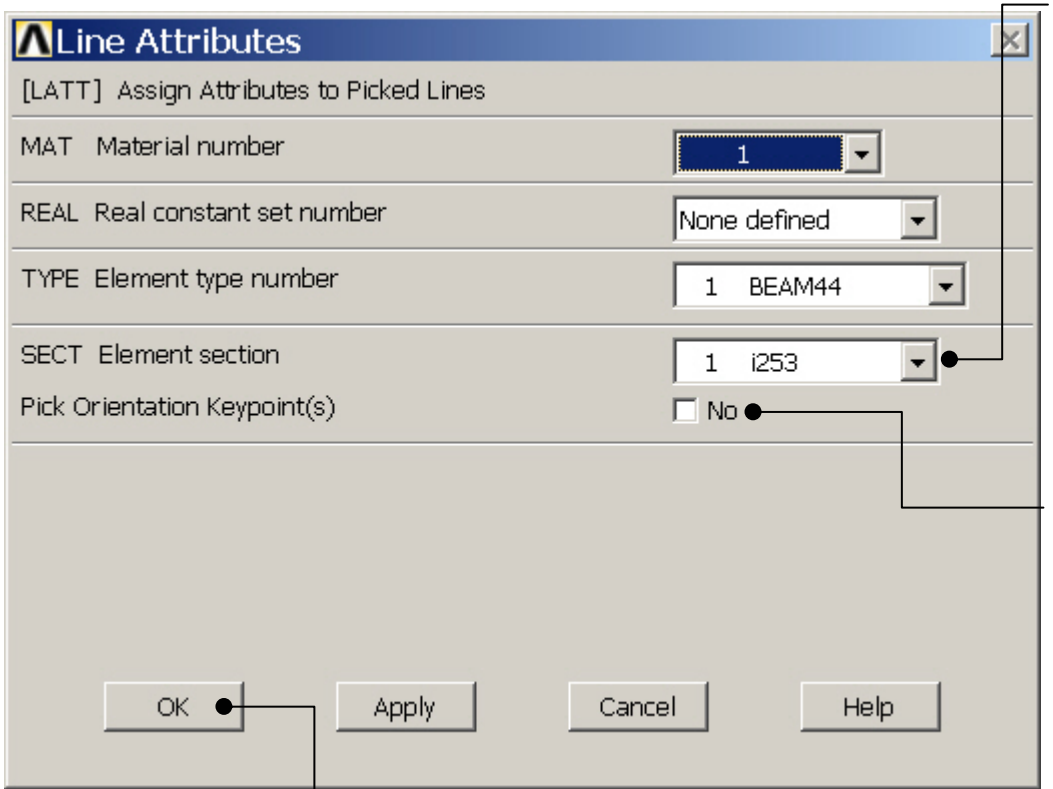
Follow the guidelines at Enter the appropiate cross-sectional data

Press OK to finish



# Example – Line Attributes

**Preprocessor > Meshing > Mesh Attributes > Picked Lines**

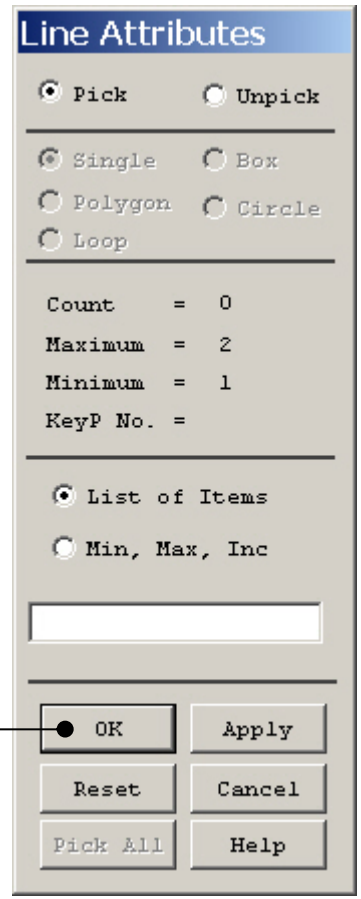



Select section I253

Change to Yes

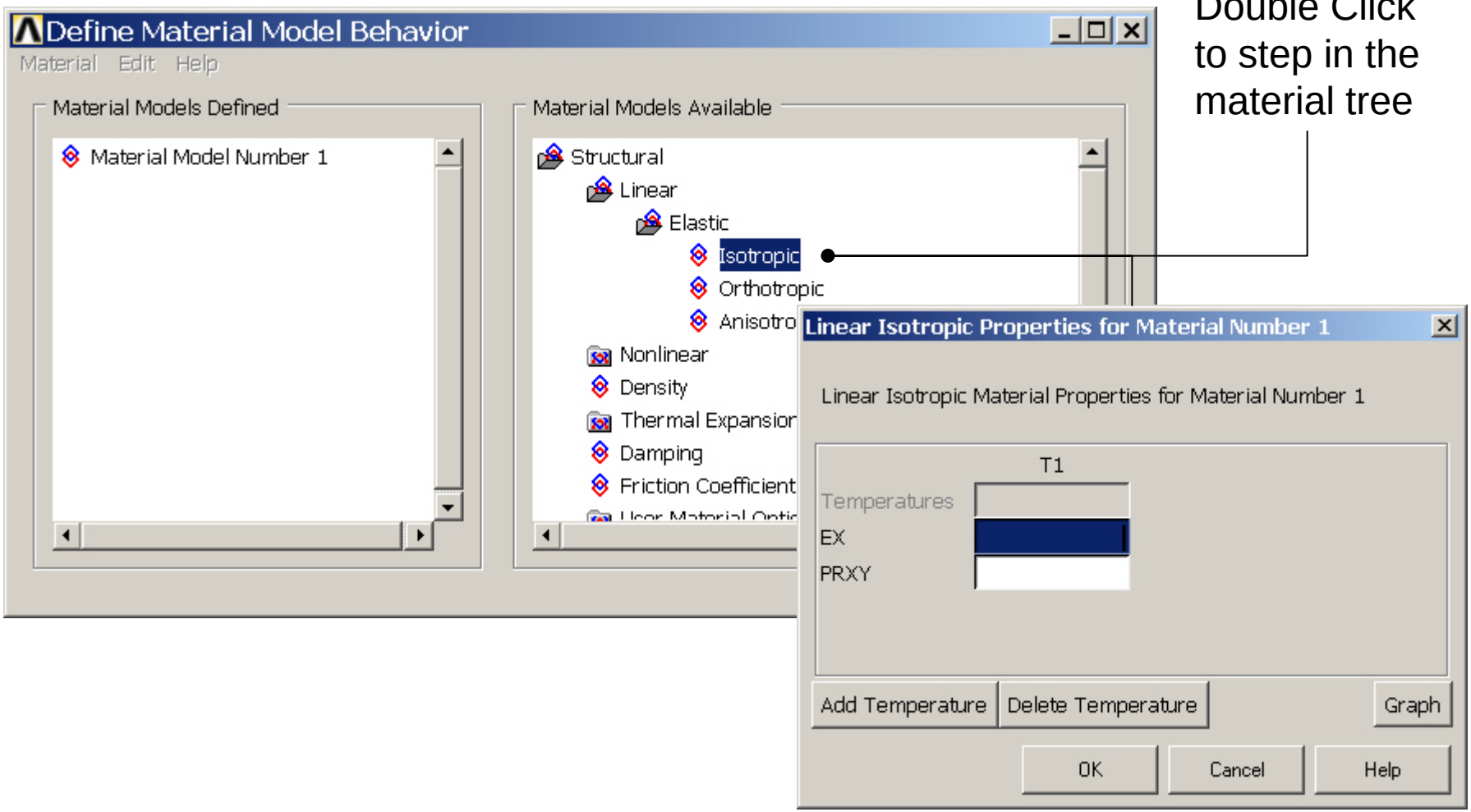
Select KP3

Press OK to finish



# Example - Material Properties

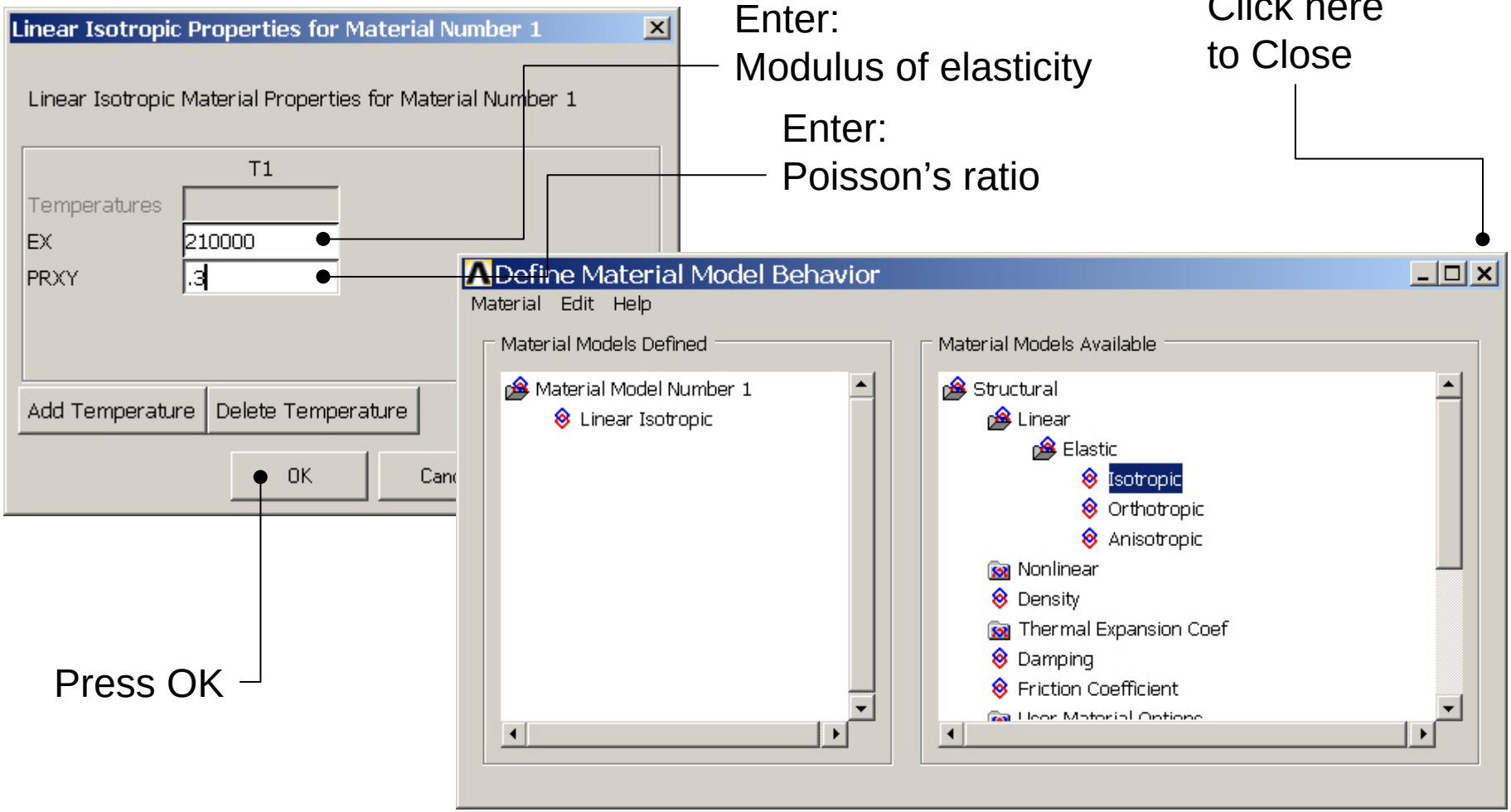
**Preprocessor > Material Props > Material Models**




Double Click to step in the material tree

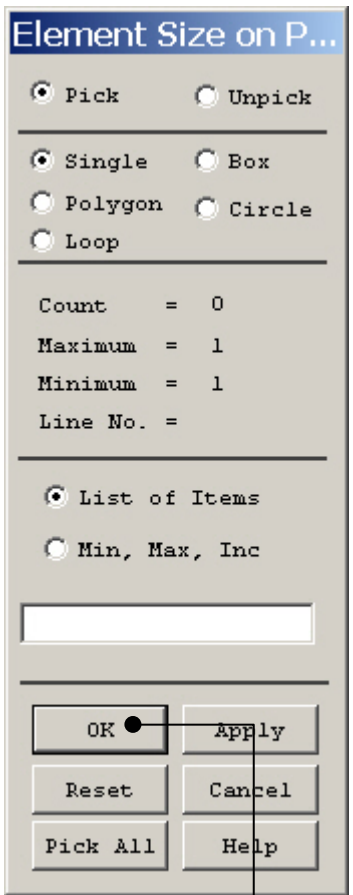
# Example - Material Properties

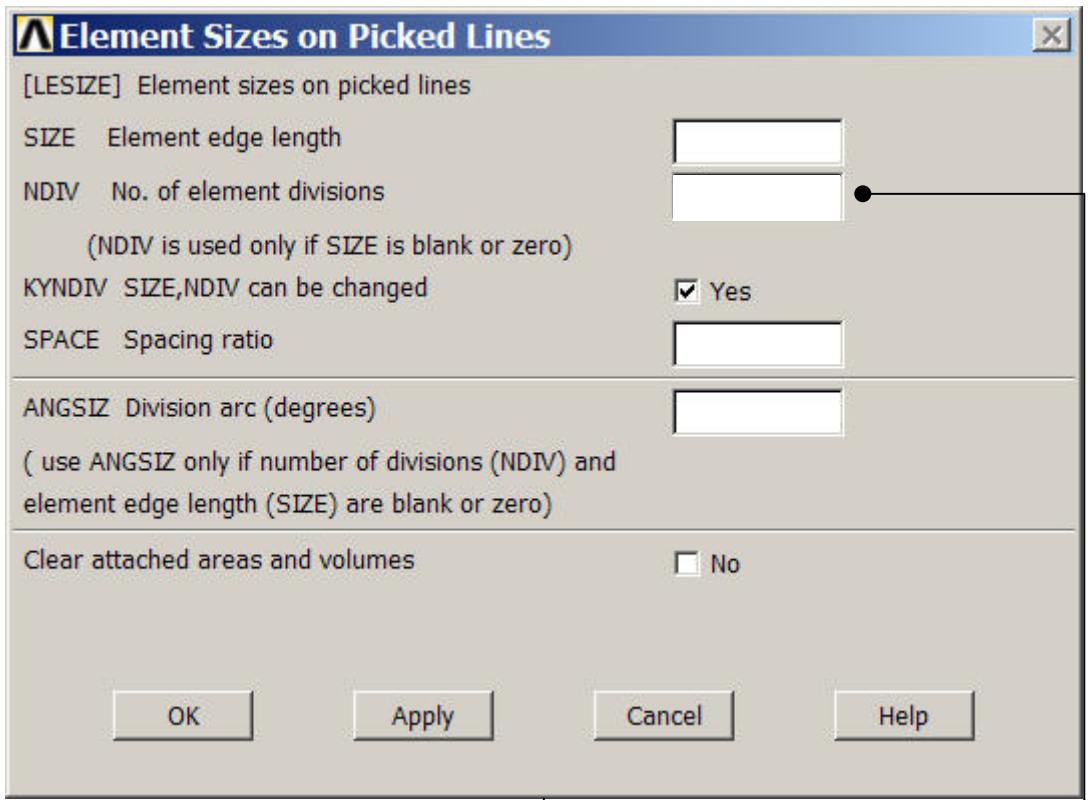
**Preprocessor > Material Props > Material Models**



# Example - Meshing

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



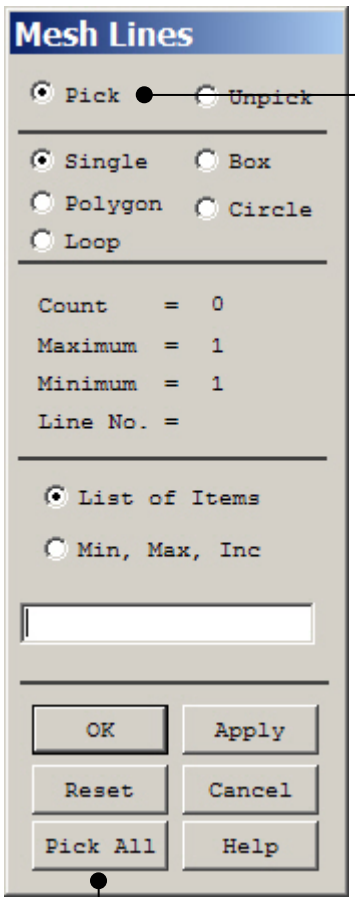

Select/Pick Lines to specify mesh size for

Press OK when finish with selection

Enter 5

# Example - Meshing

**Preprocessor > Meshing > Mesh > Lines**

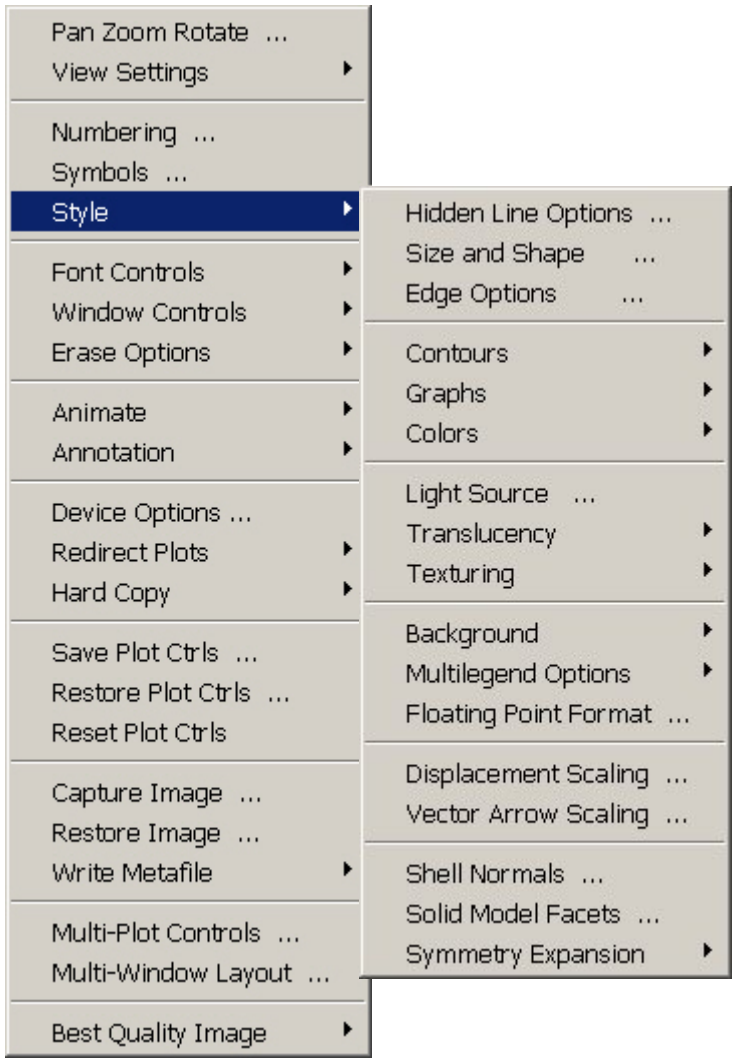


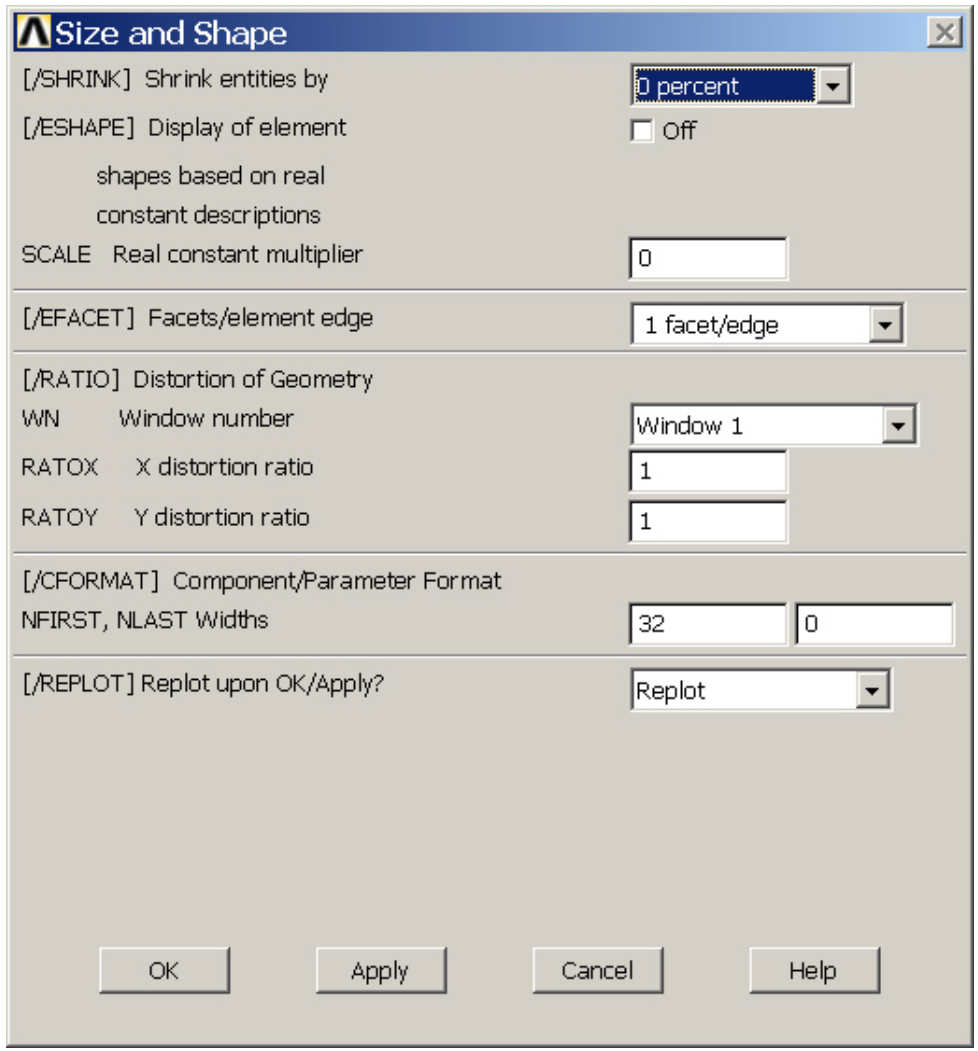

Select individual lines to be meshed by Picking

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

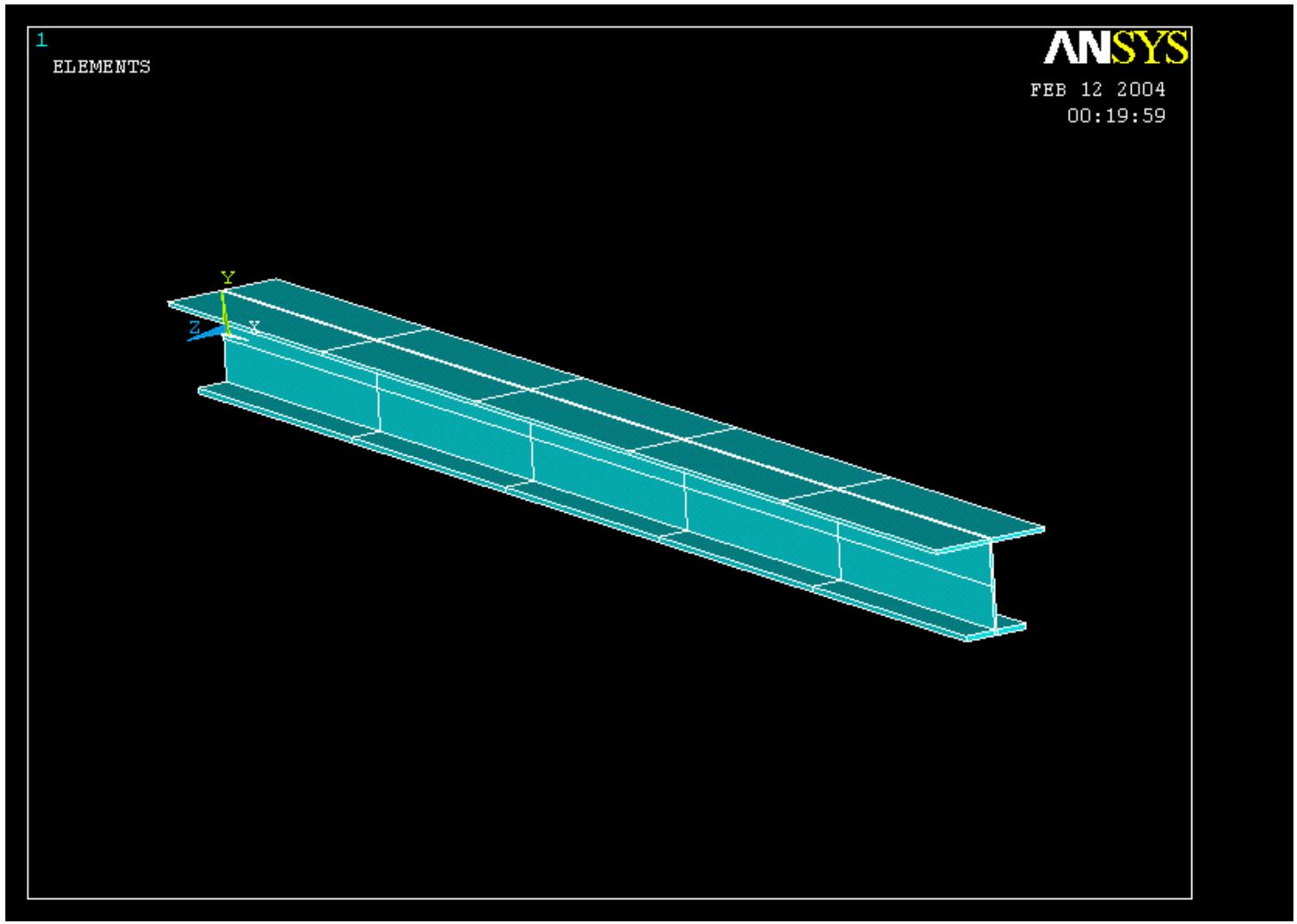
Select all lines defined to be meshed

# Example - PlotCtrls Menu



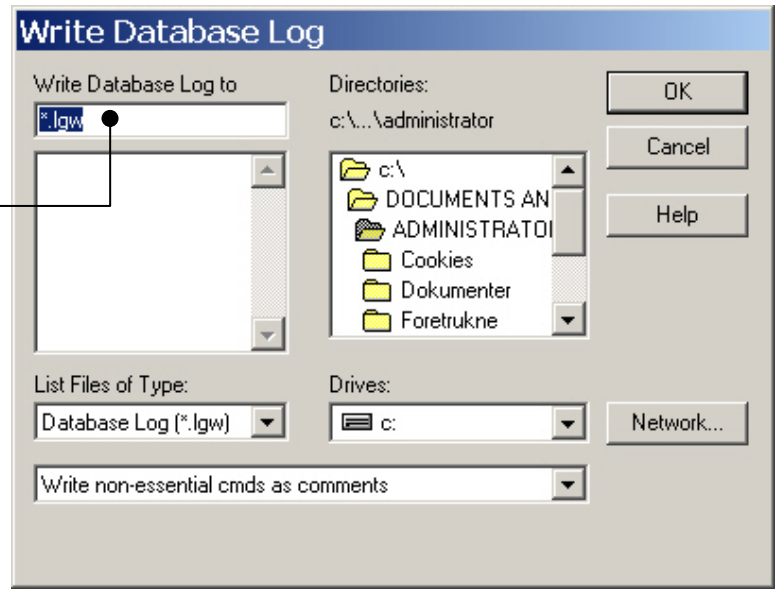
# Example – Display of Element



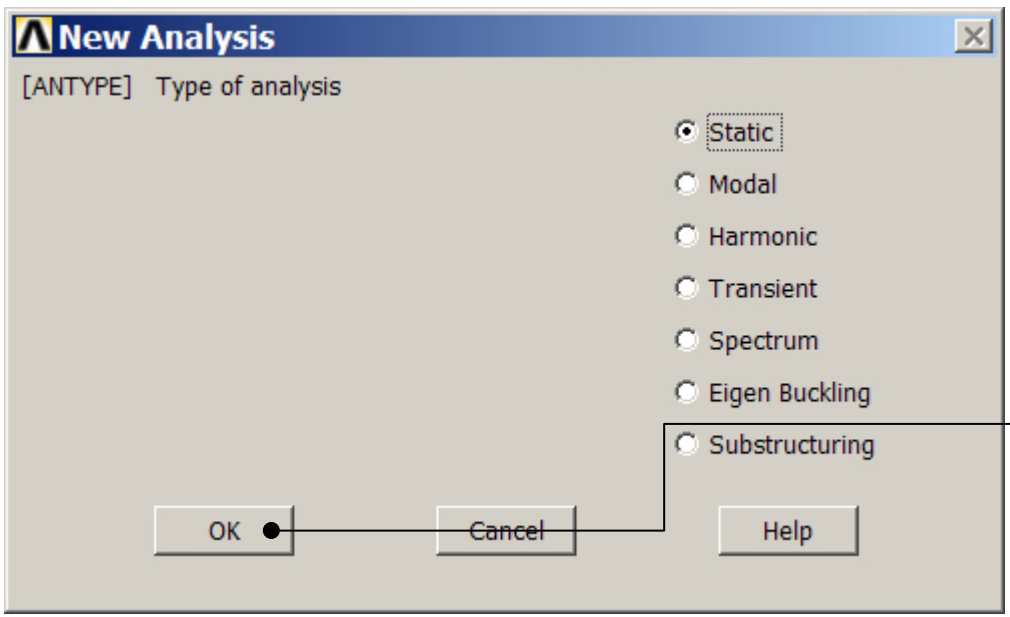
# Example – Analysis Type

**File > Write DB log file**

Enter “example0504.lgw”

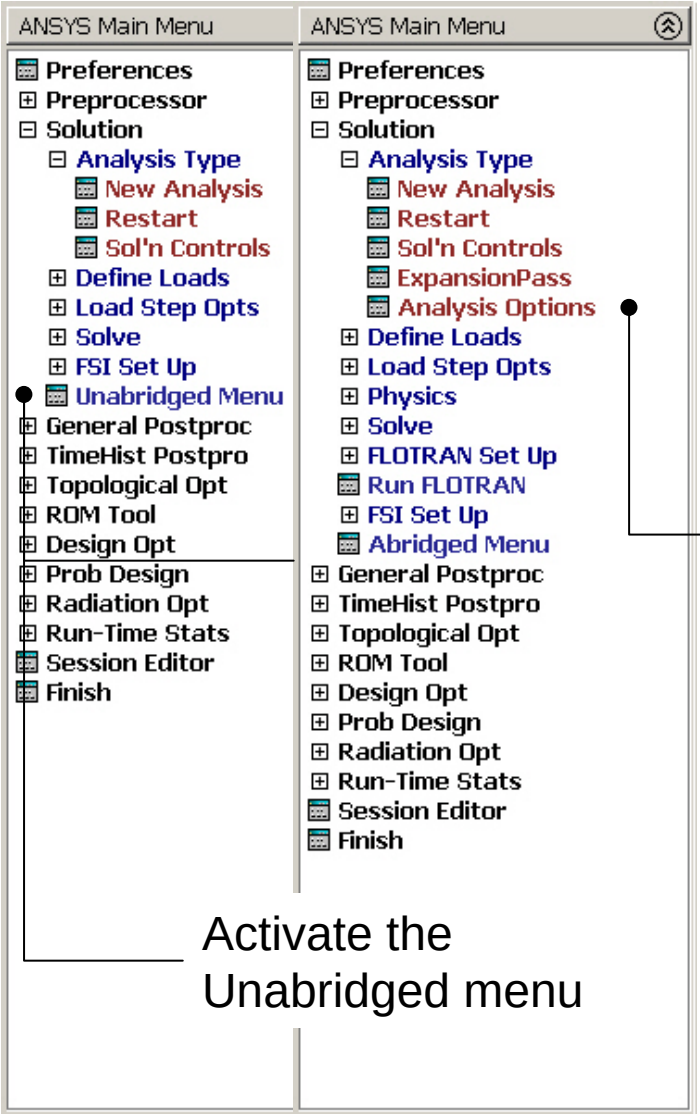



**Solution > Analysis Type > New Analysis**

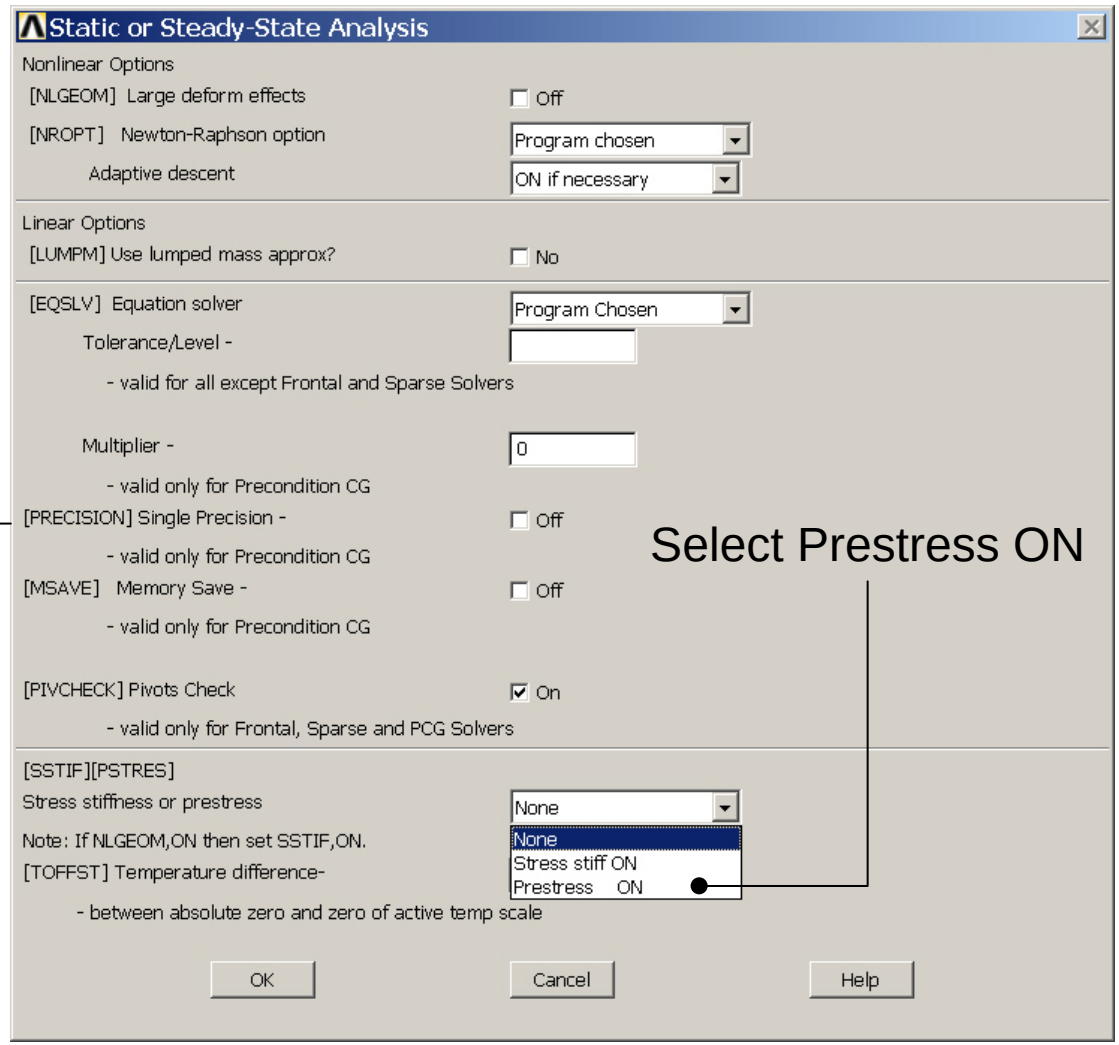



Press OK

# Static solution – Analysis Options




Activate the Unabridged menu

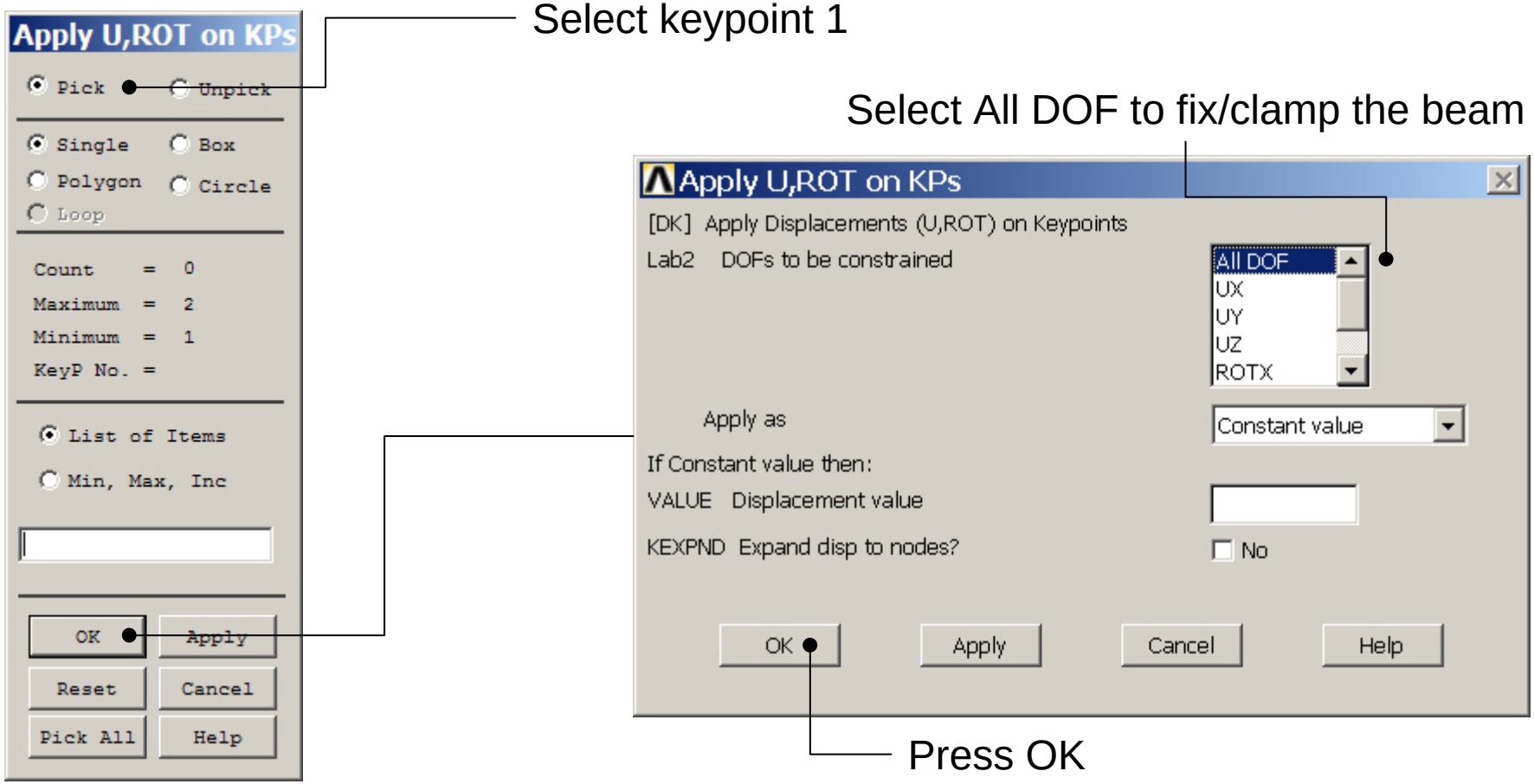



Select Prestress ON

Example0504

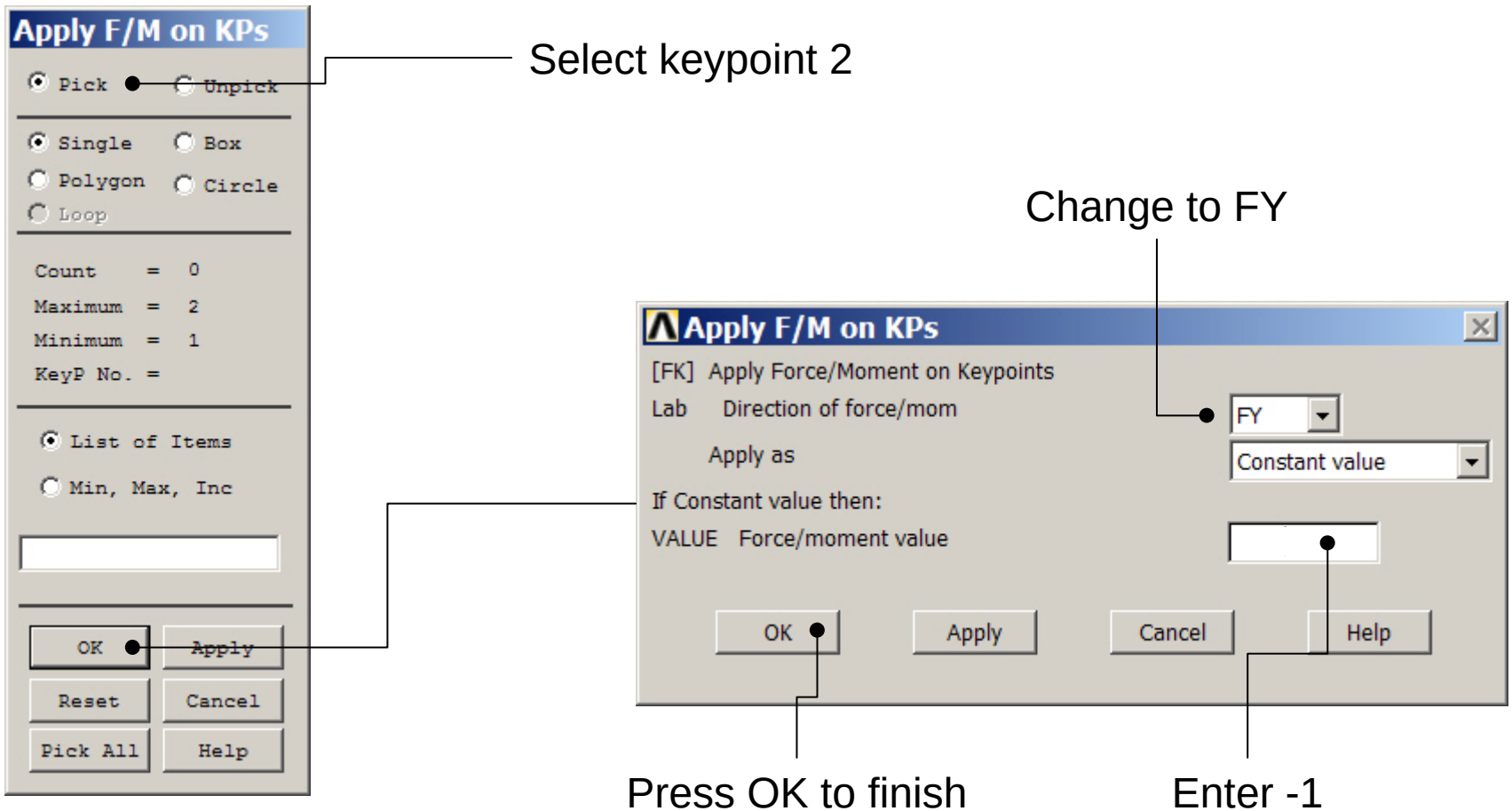
# Example – Define Loads

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

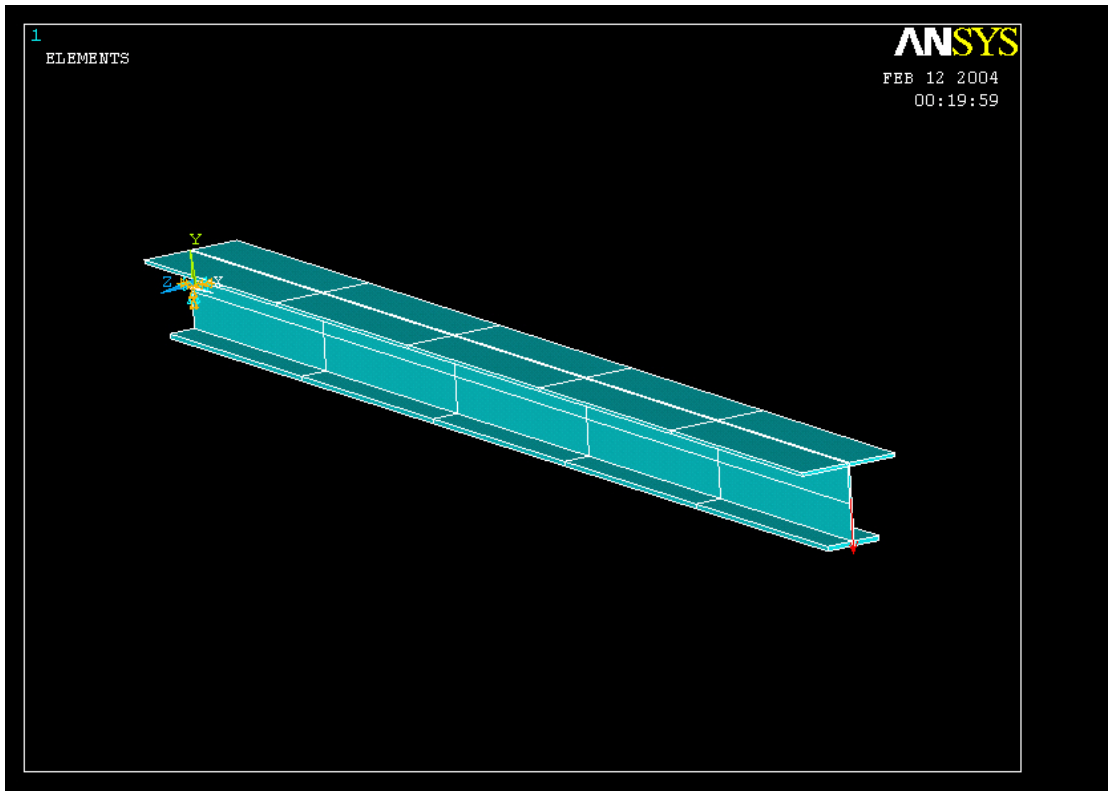


# Example – Define Loads

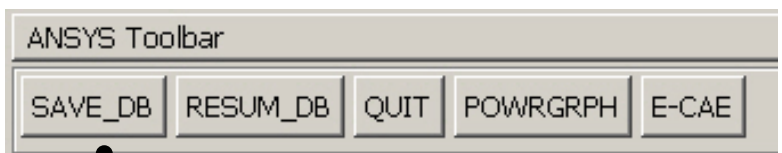
**Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints**



# Example - Save



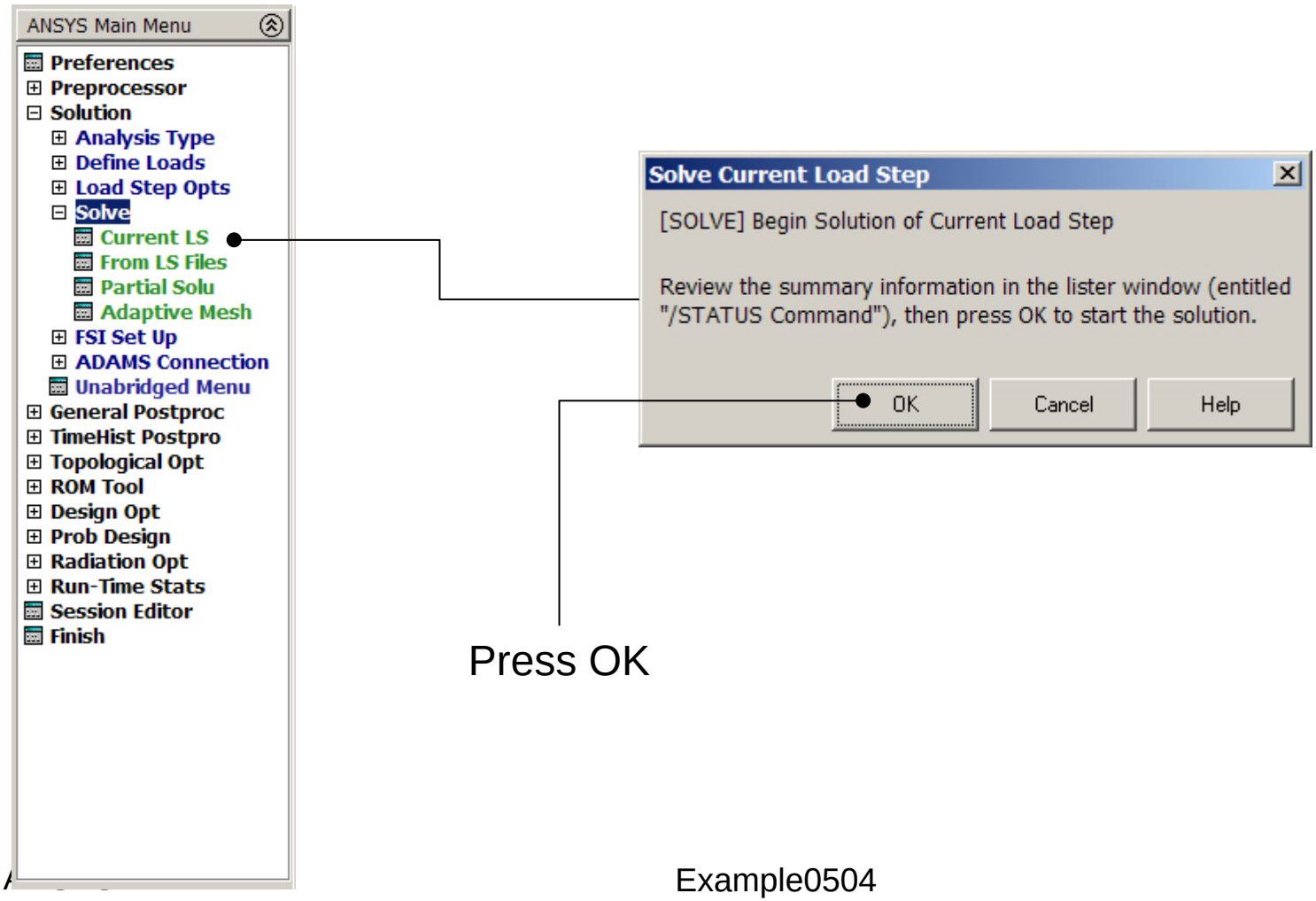

Display of Analysis model




Save the model

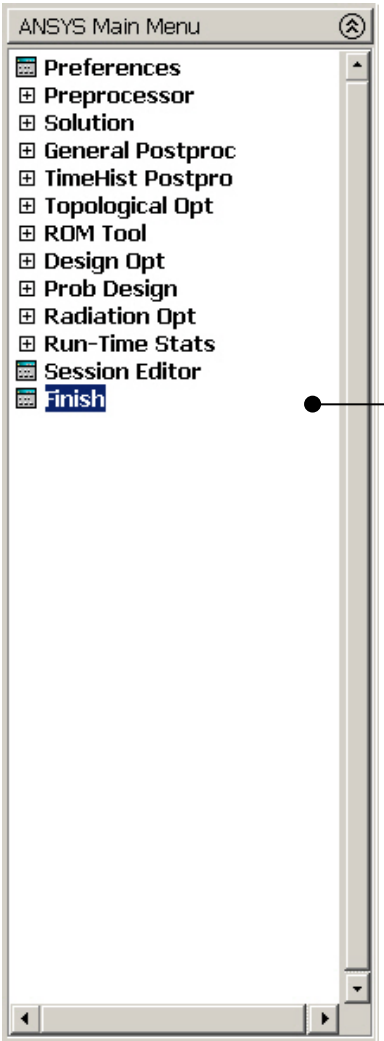
# Example - Solve

**Solution > Solve > Current LS**




Press OK

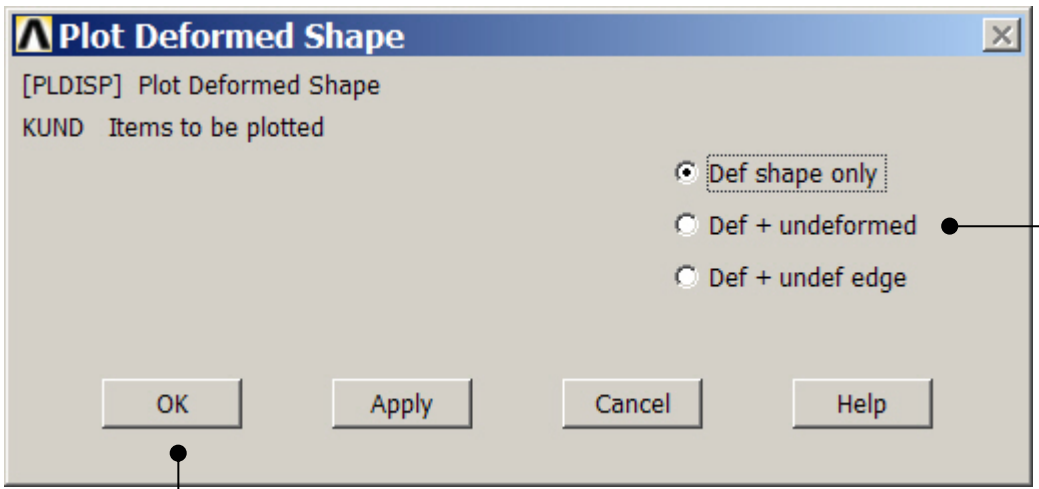
# Example - Finish




Press Finish to end the static solution

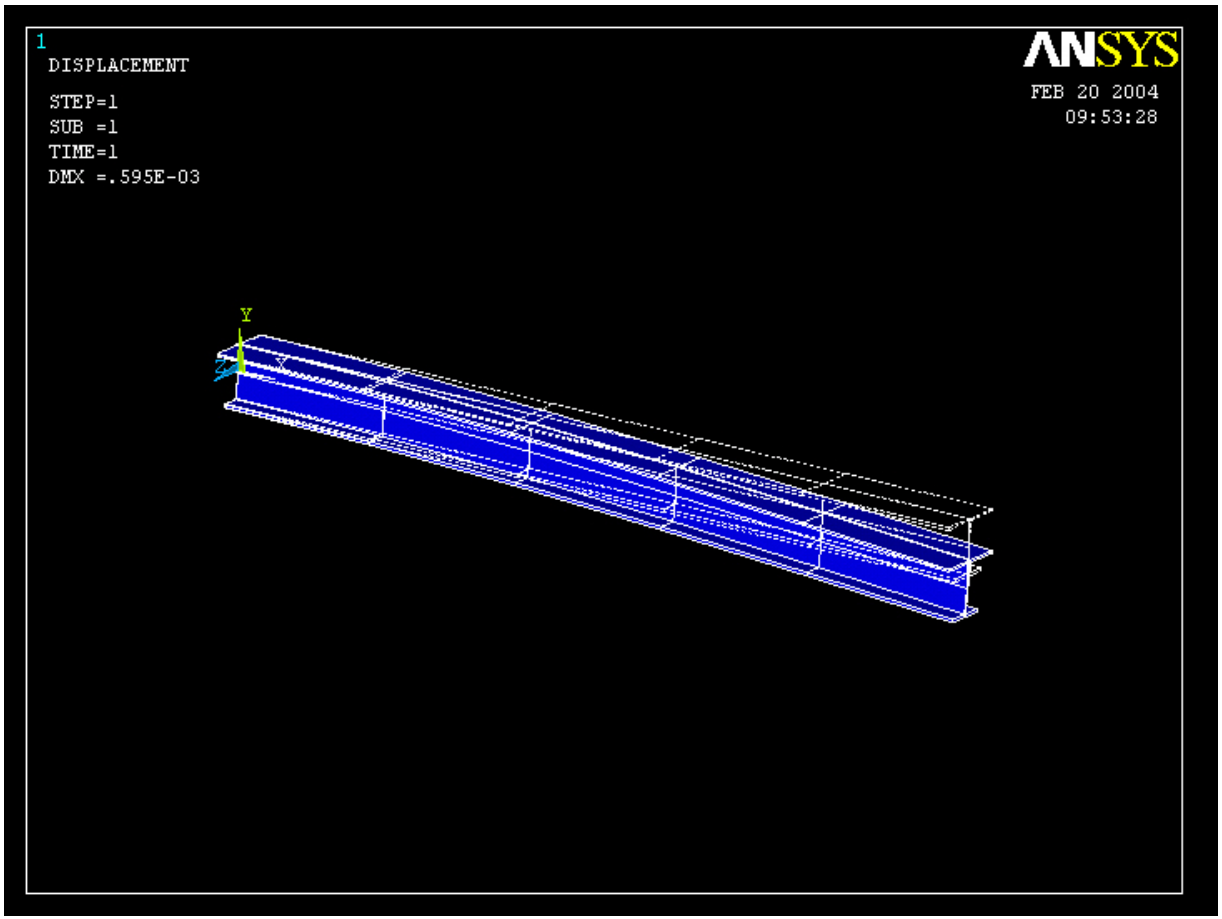
# Example - PostProcessing

**General Postproc > Plot Results > Deformed Shape**




Select “Def+undeformed” and Press OK

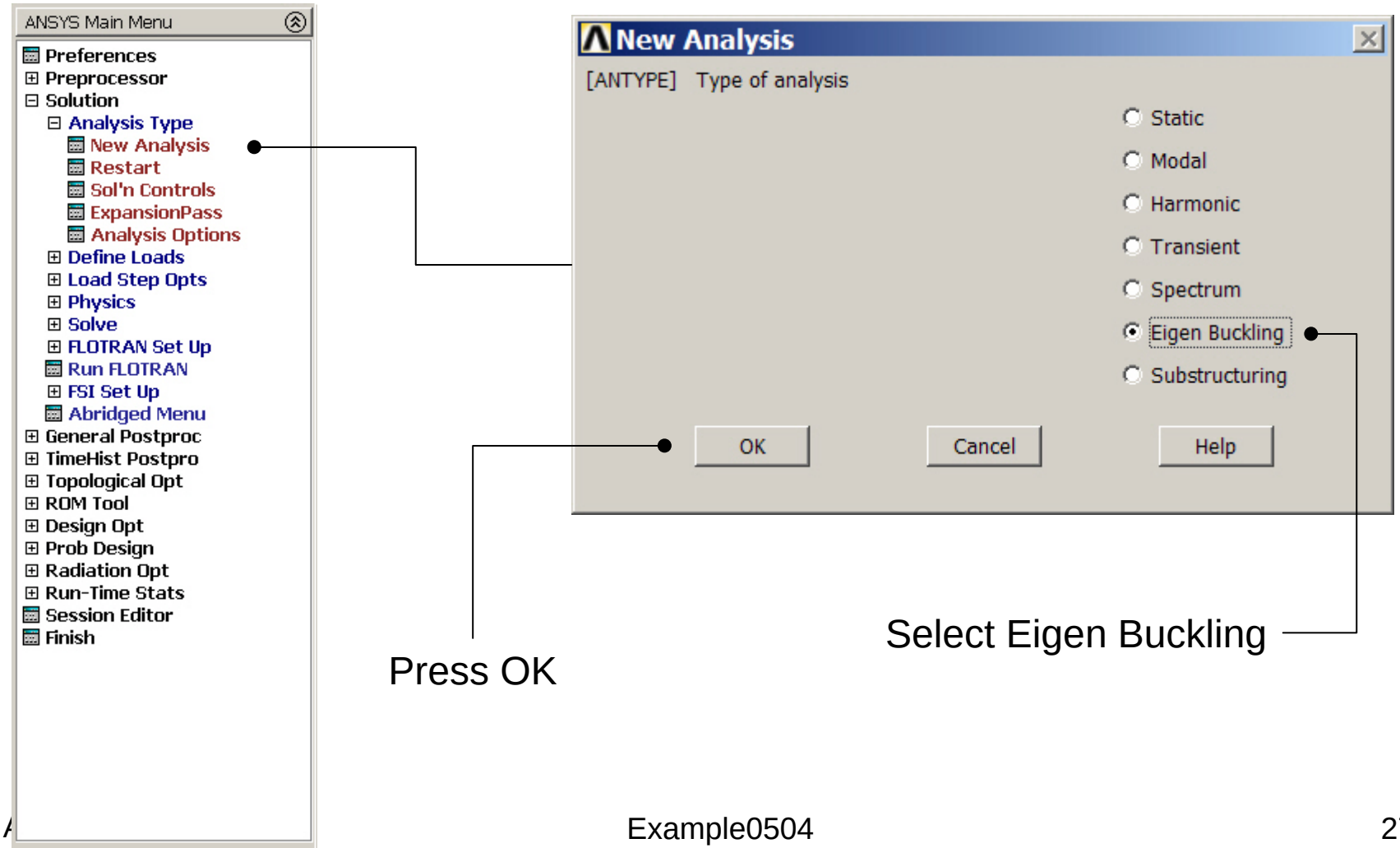
# Example - PostProcessing




Read Maximum displacement: DMX

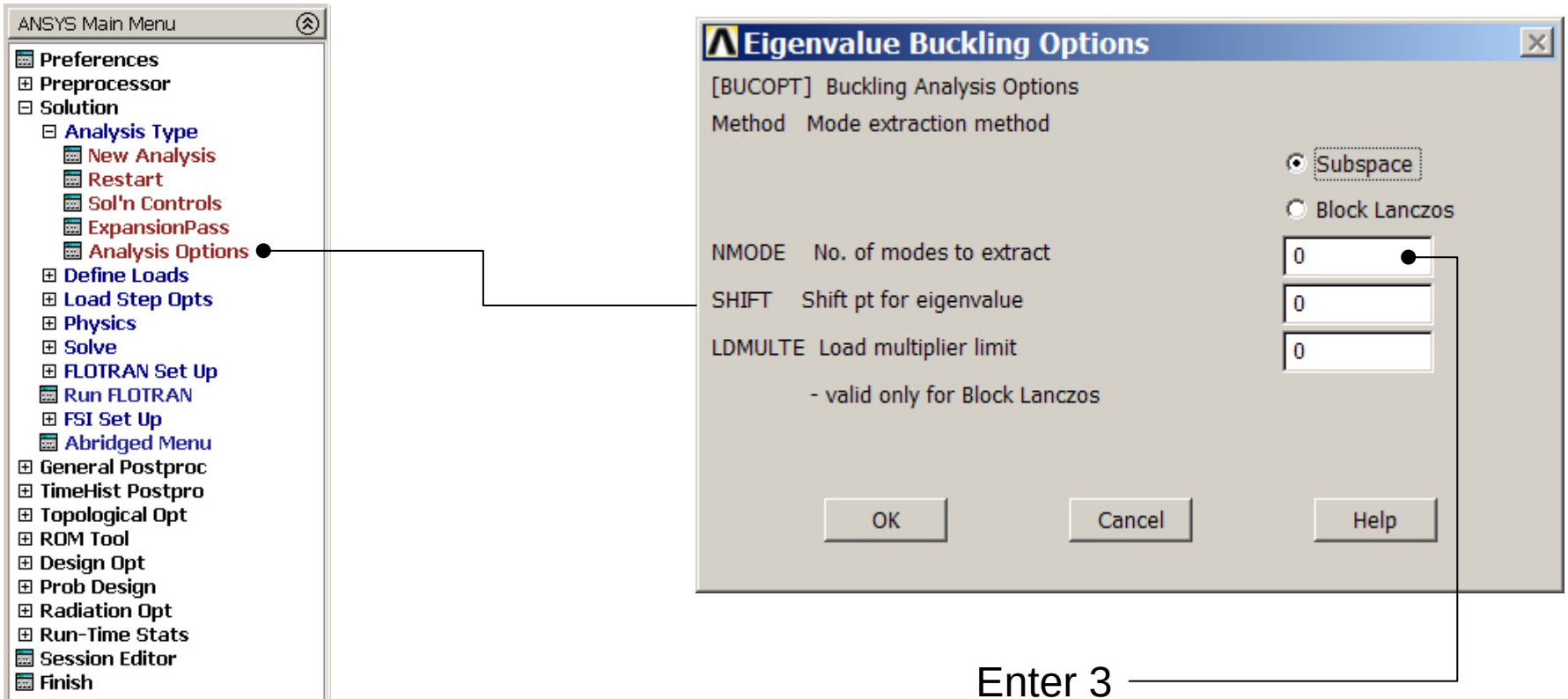
# Eigen Buckling - New Analysis

**Main Menu> Solution> Analysis Type> New Analysis**



# Eigen Buckling – Analysis Options

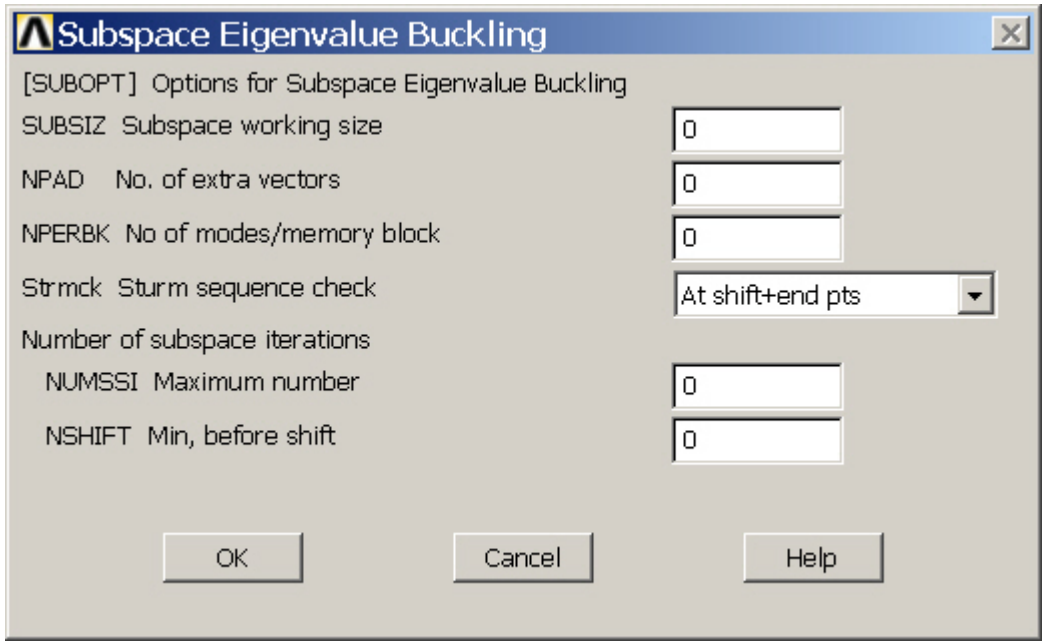
**Main Menu> Solution> Analysis Type> Analysis Options**



# 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.

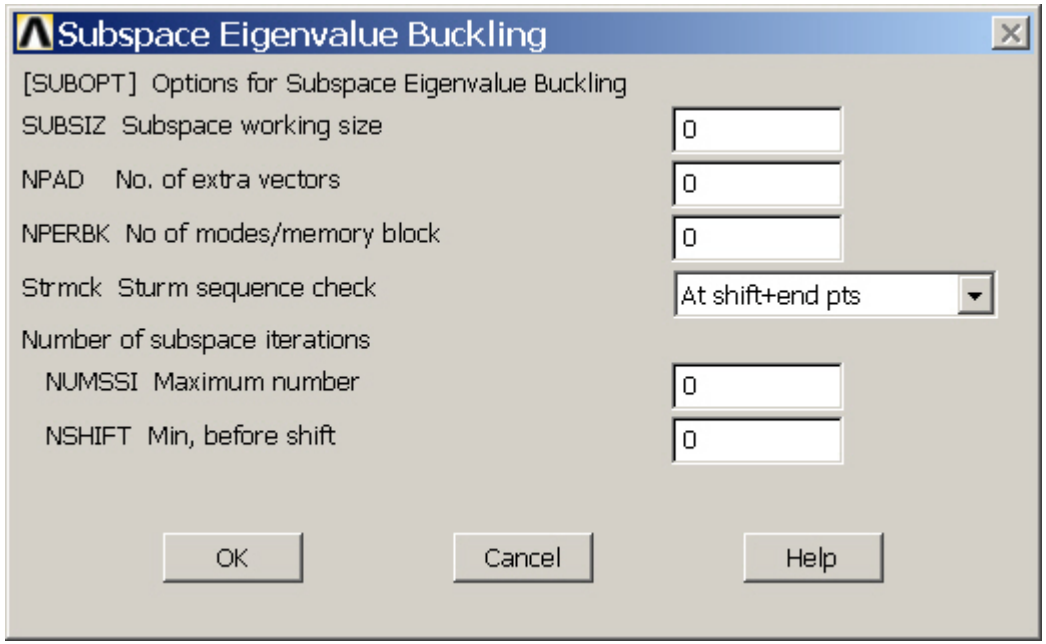
# Example – Subspace Options




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).

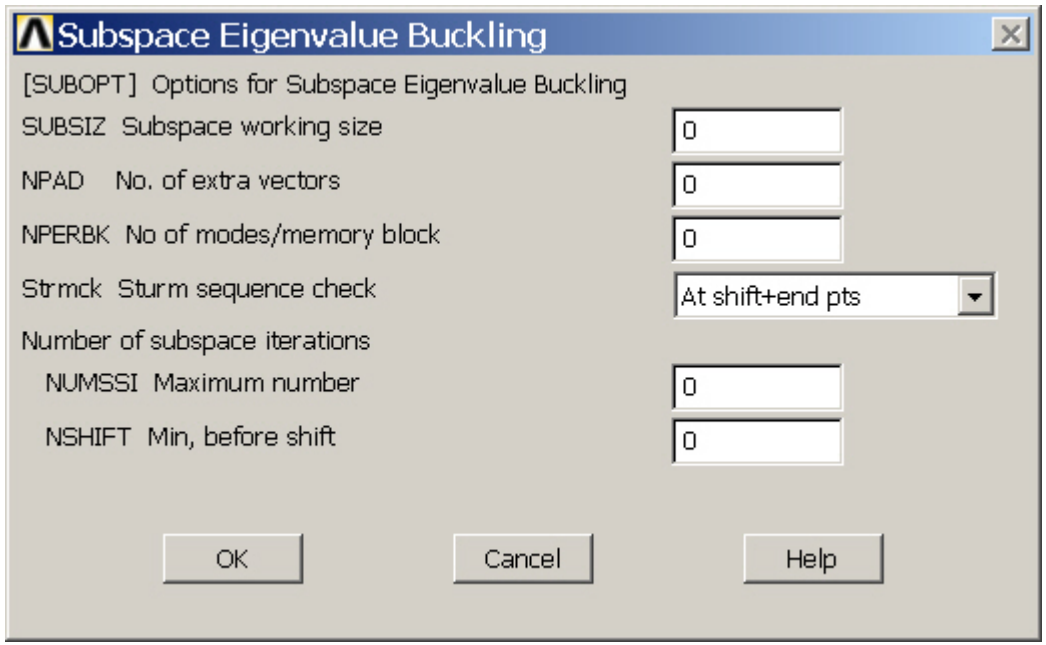
Number of extra vectors used in the iterations. Defaults to 4. The total number of vectors used is *NMODE* +*NPAD*.

# 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.

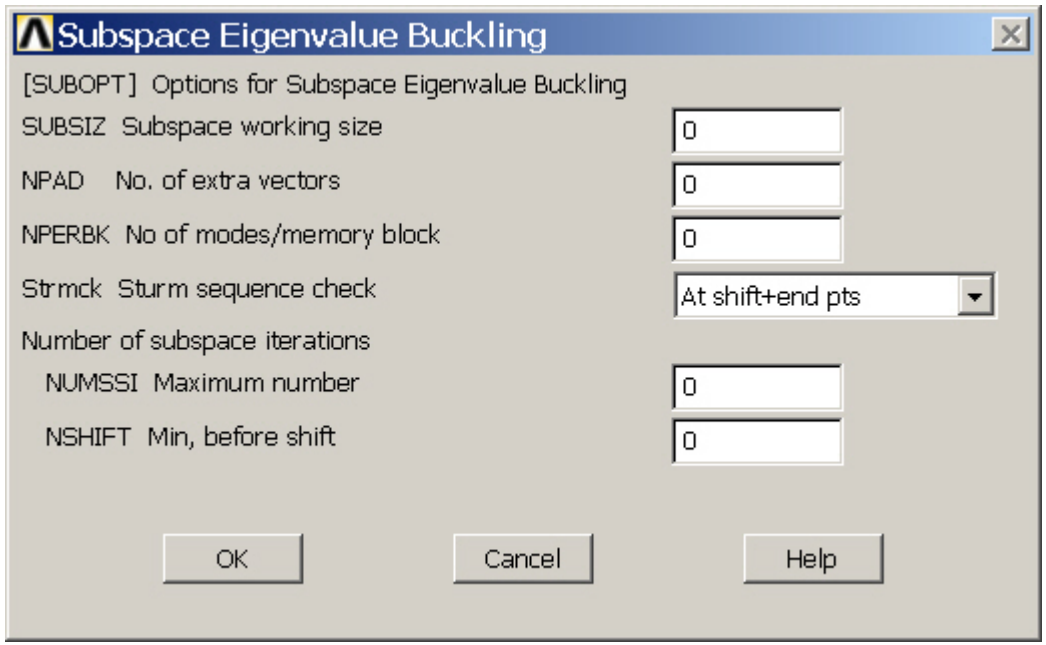
# Example – Subspace Options




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




Sturm sequence check key:

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

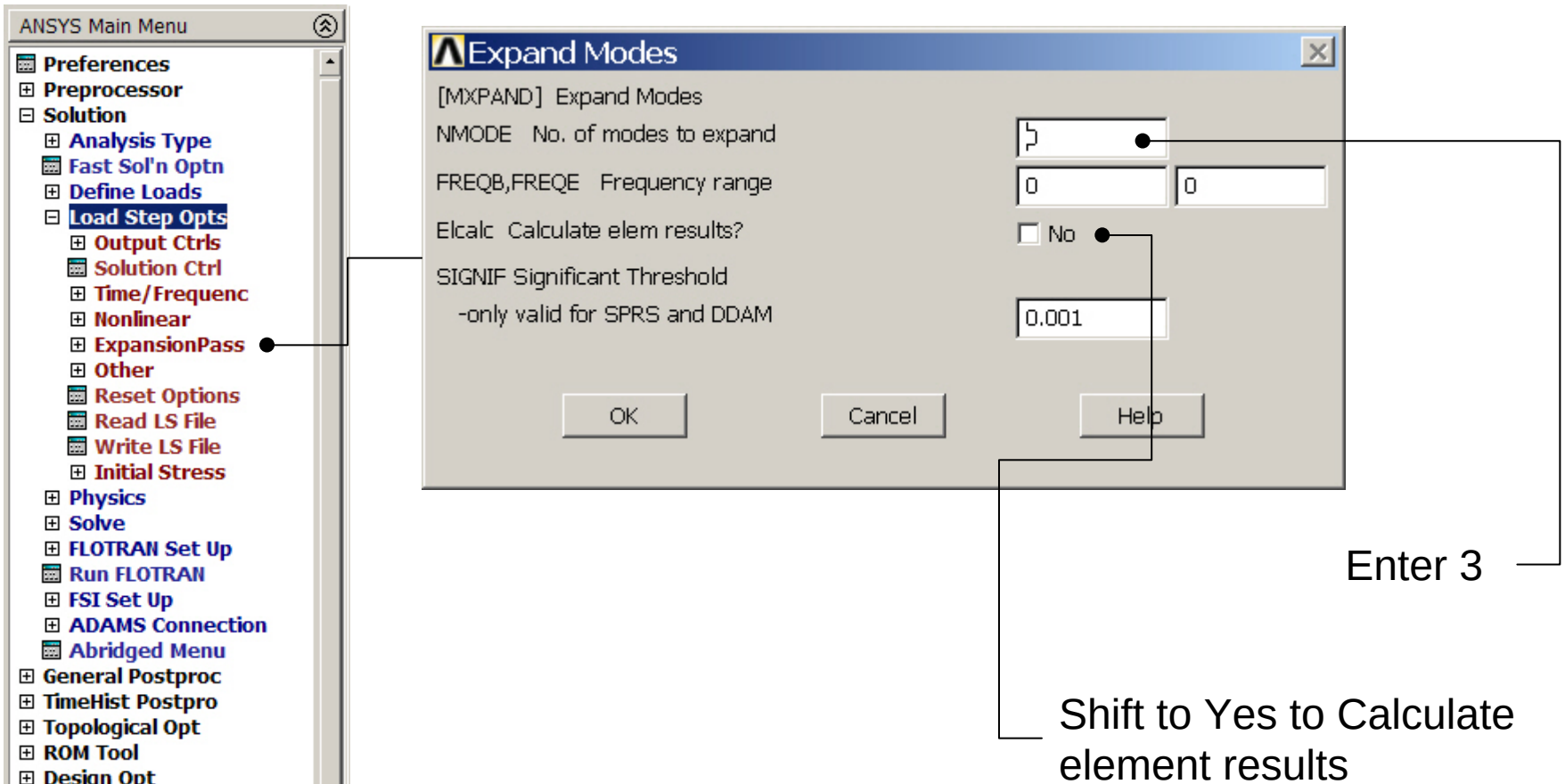
PART --
Perform check only at all shift points.

NONE --
Do not perform Sturm sequence check.

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.

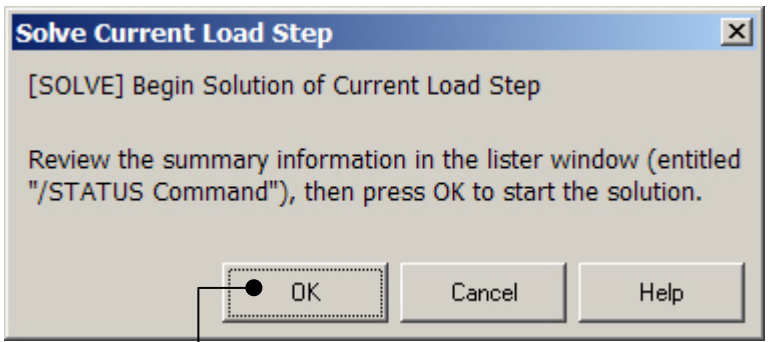
# Eigen Buckling – Expanding Modes

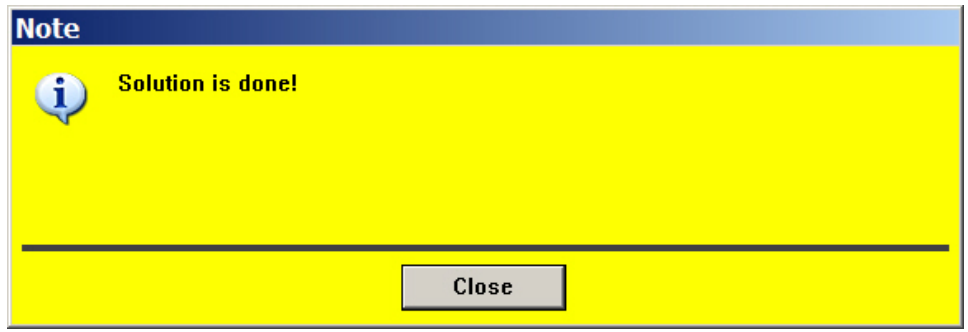
**Main Menu> Solution> Load Step Opts > ExpansionPass > Single Expand > Expand Modes**



# Example - Solve

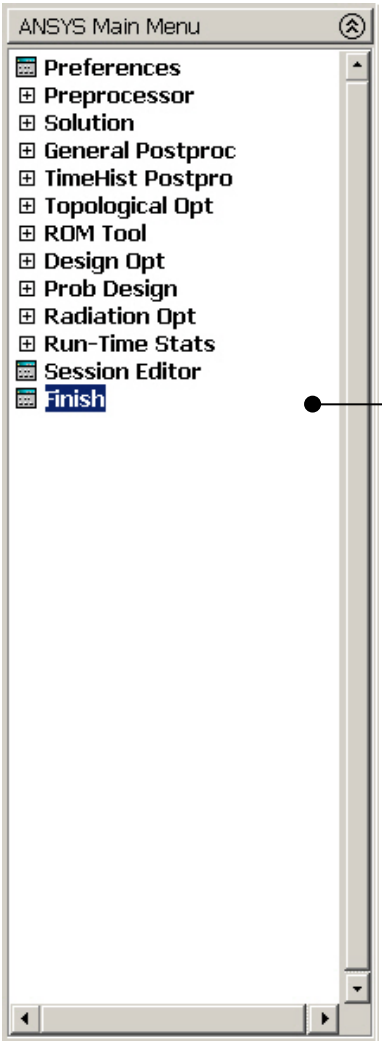
**Solution > Solve > Current LS**



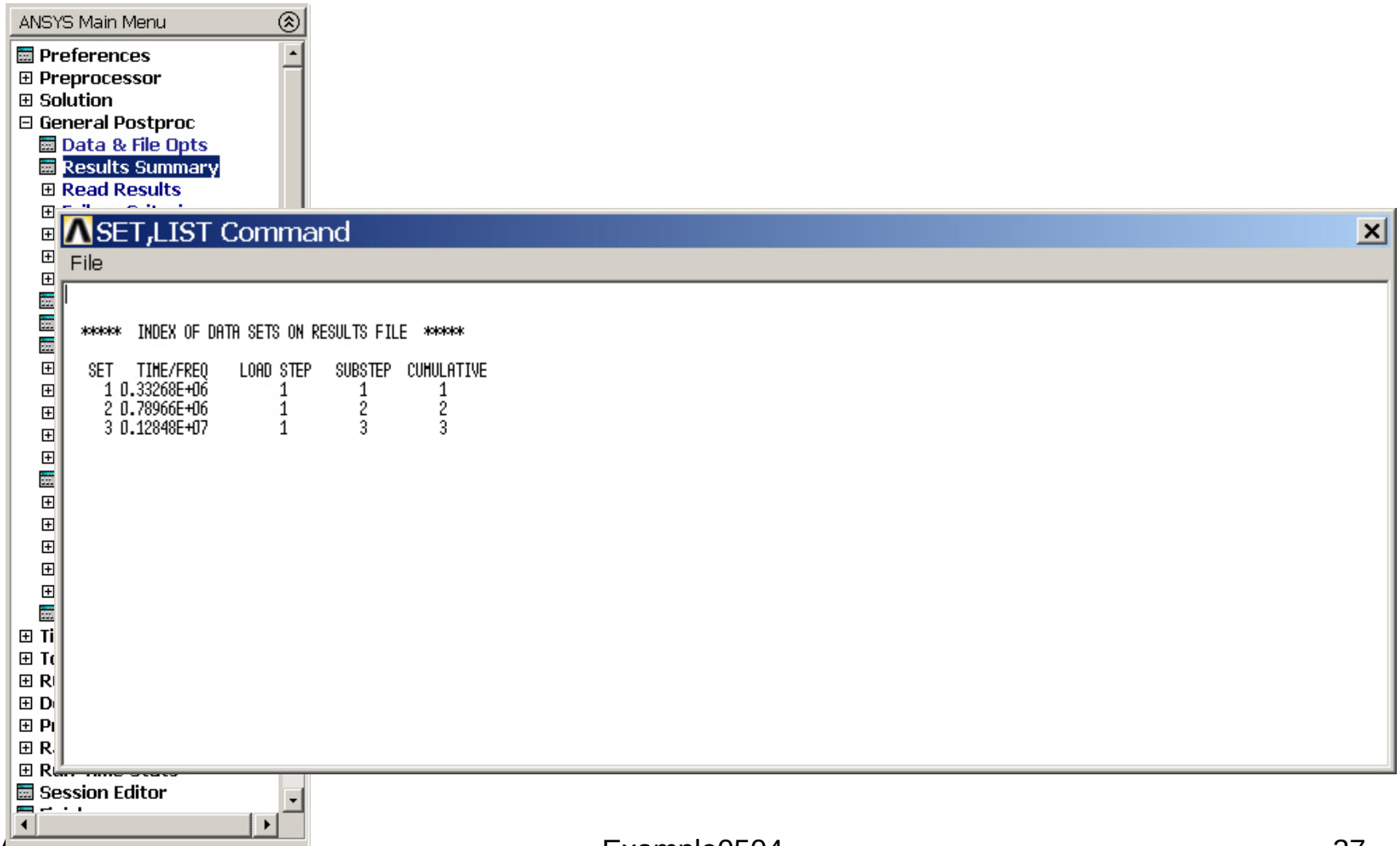

Press OK

# Example - Finish

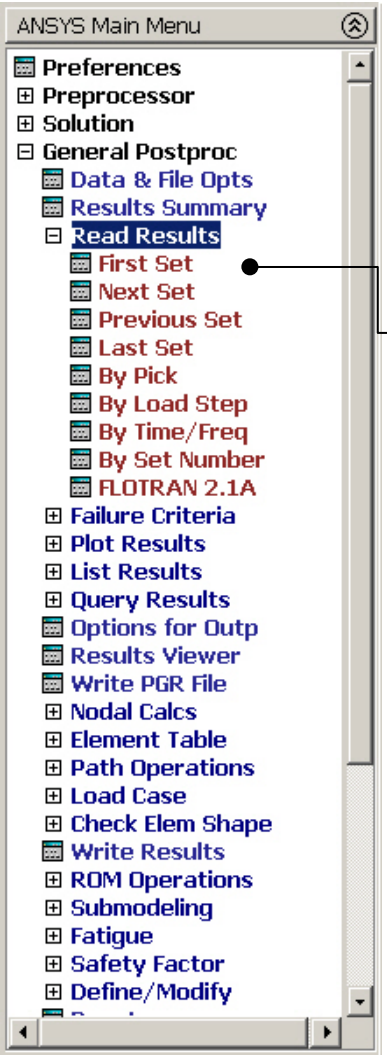



Press Finish to end the eigen buckling solution

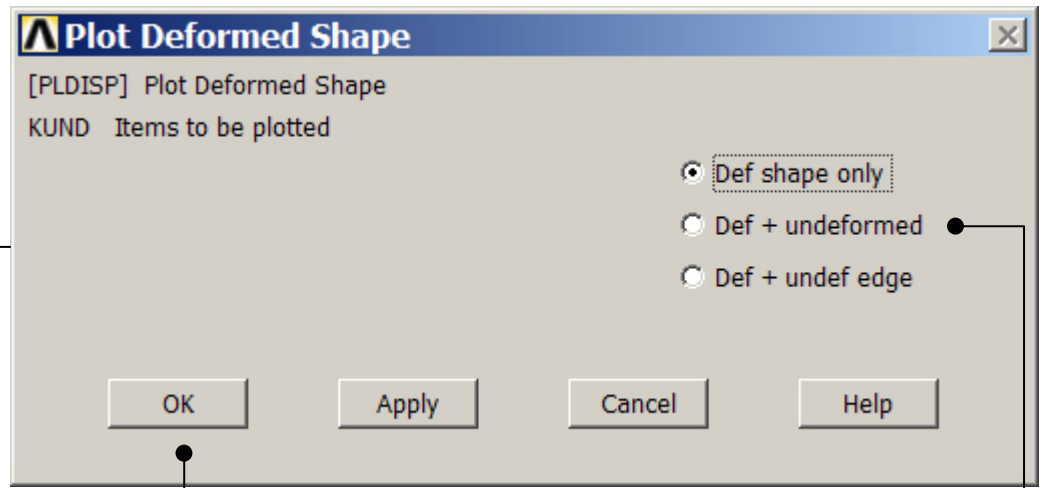
# Example – Results Summary



# Example – Read Results

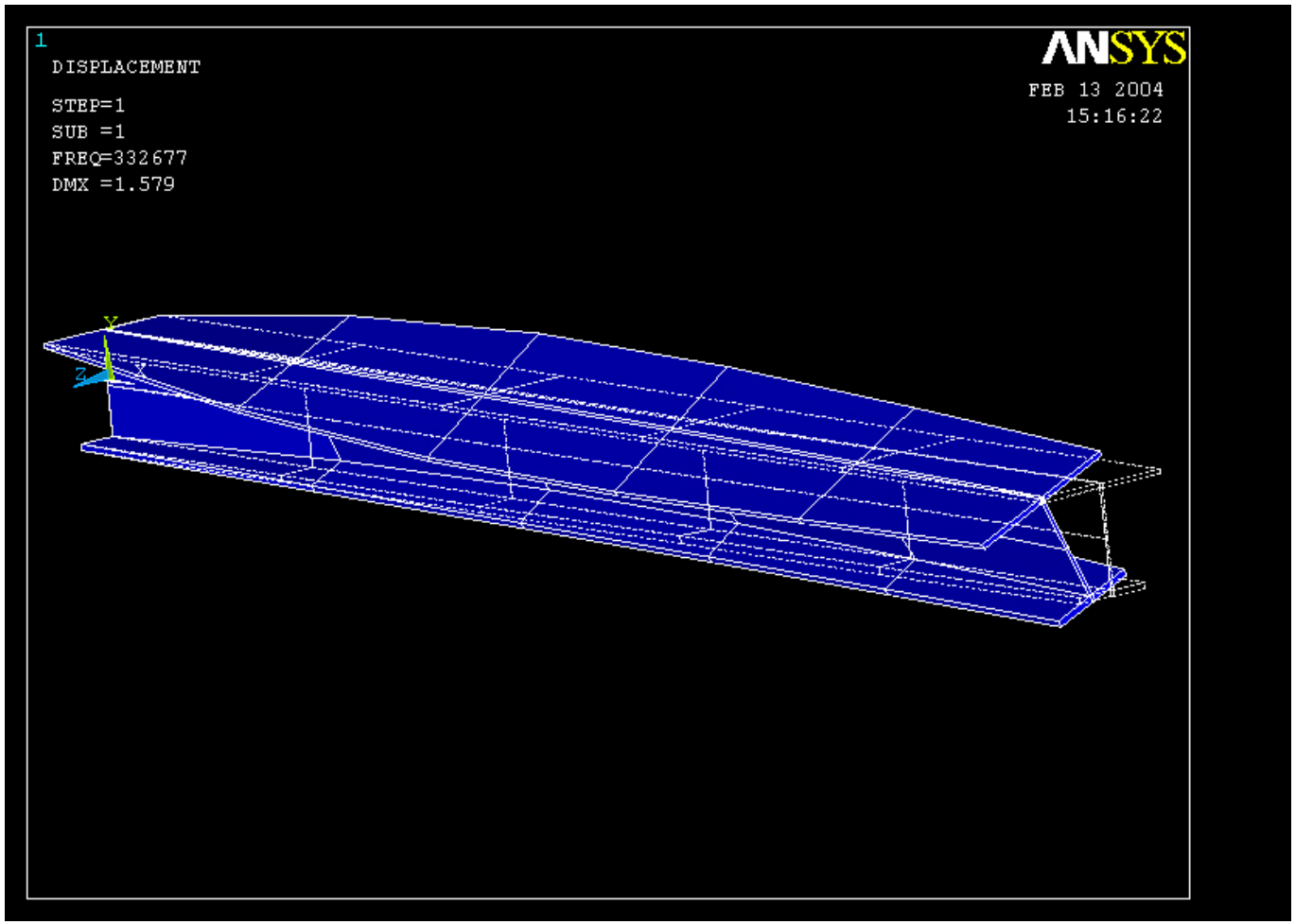



**General Postproc > Plot Results > Deformed Shape**

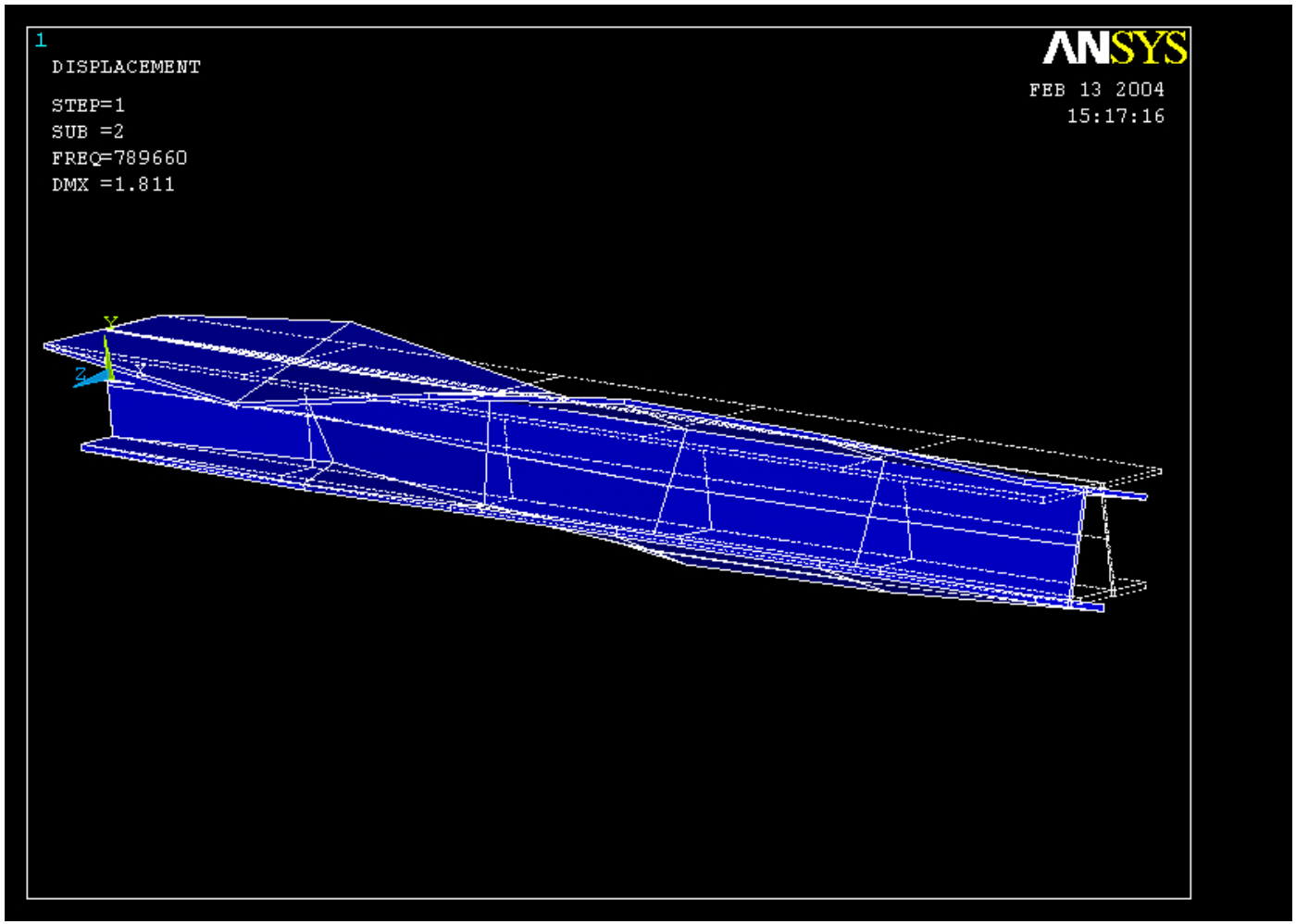



Select “Def+undeformed” and Press OK

# Example – First set



# Example – Next set



# Example – Next set