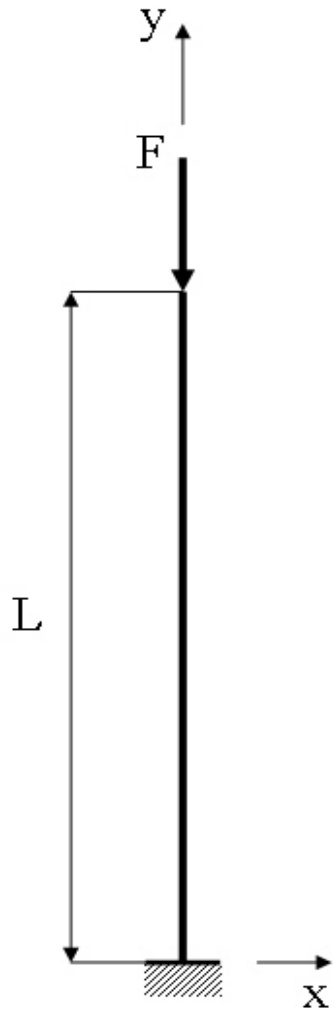


# Course in ANSYS

Example0500

# Example – Column beam



## Objective:

Compute the critical buckling load and display the mode shape

## Tasks:

Create a table and compare results with results obtained from buckling theory?

Display the deflection figure?

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

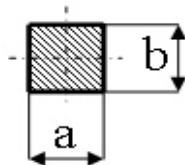
$$\nu = 0.3$$

$$L = 1000 \text{ mm}$$

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

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

$$F = ?$$



# Modelling considerations

- As you begin your model generation, you will (consciously or unconsciously) make a number of decisions that determine how you will mathematically simulate the physical system:
  - What are the objectives of your analysis?
  - Will you model all, or just a portion, of the physical system?
  - How much detail will you include in your model?
  - What kinds of elements will you use? How dense should your finite element mesh be?
- In general, you will attempt to balance computational expense (CPU time, etc.) against precision of results as you answer these questions.
- The decisions you make in the planning stage of your analysis will largely govern the success or failure of your analysis efforts.

# Modelling considerations

- Linear or Higher Order Elements
- Take Advantage of Symmetry
  - The axis of symmetry *must* coincide with the global Cartesian Y-axis.
  - Negative nodal X-coordinates are not permitted.
  - The global Cartesian Y-direction represents the axial direction, the global Cartesian X-direction represents the radial direction, and the global Cartesian Z-direction corresponds to the circumferential direction.
  - Your model should be assembled using appropriate element types:
    - For axisymmetric models, use applicable 2-D solids with KEYOPT(3) = 1, and/or axisymmetric shells. In addition, various link, contact, combination, and surface elements can be included in a model that also contains axisymmetric solids or shells. (The program will not realize that these "other" elements are axisymmetric unless axisymmetric solids or shells are present.)
- How Much Detail to Include
- Appropriate Mesh Density

# Example - title

**Utility Menu > File > Change Jobname**

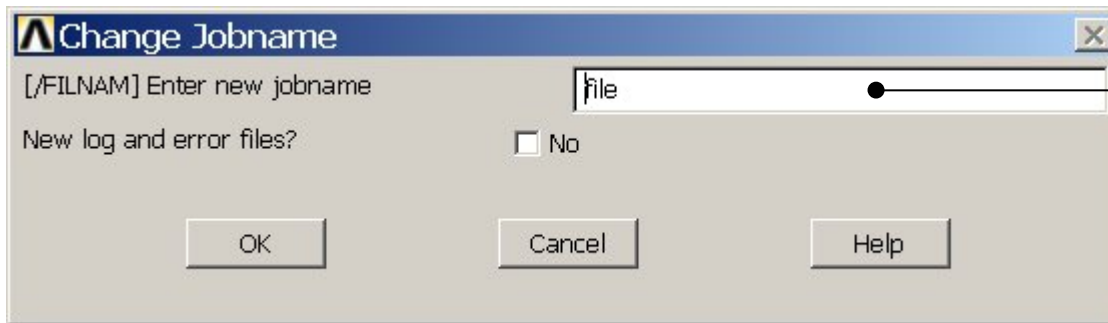


GUI

/jobname, Example0500



Command line entry

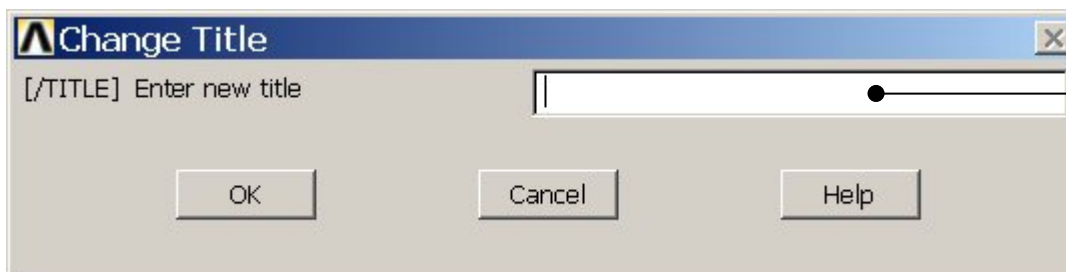


Enter: Example0500

**Utility Menu > File > Change Title**

/title, Column beam

Enter: Column beam



# Example - Keypoints

Note: An empty # result in automatic numbering.

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

/PREP7

K,,,

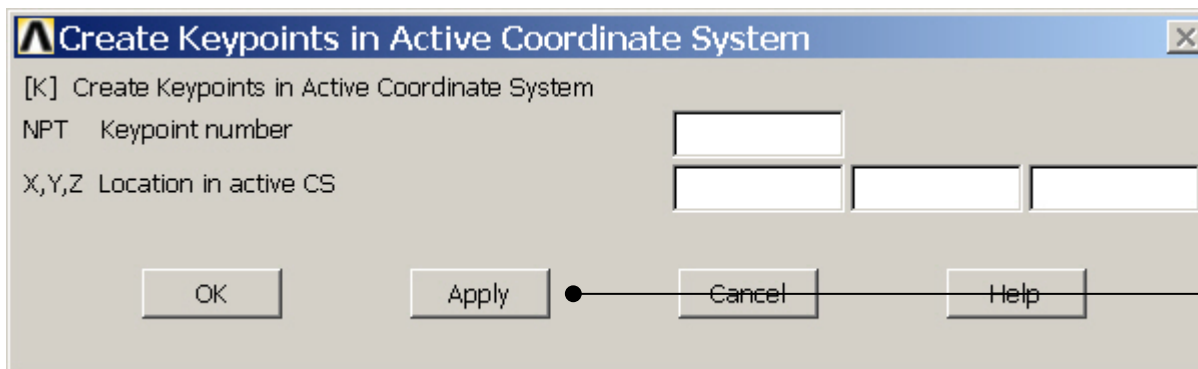
K,,,1000,

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 0,1000,0 and  
Press **Apply**

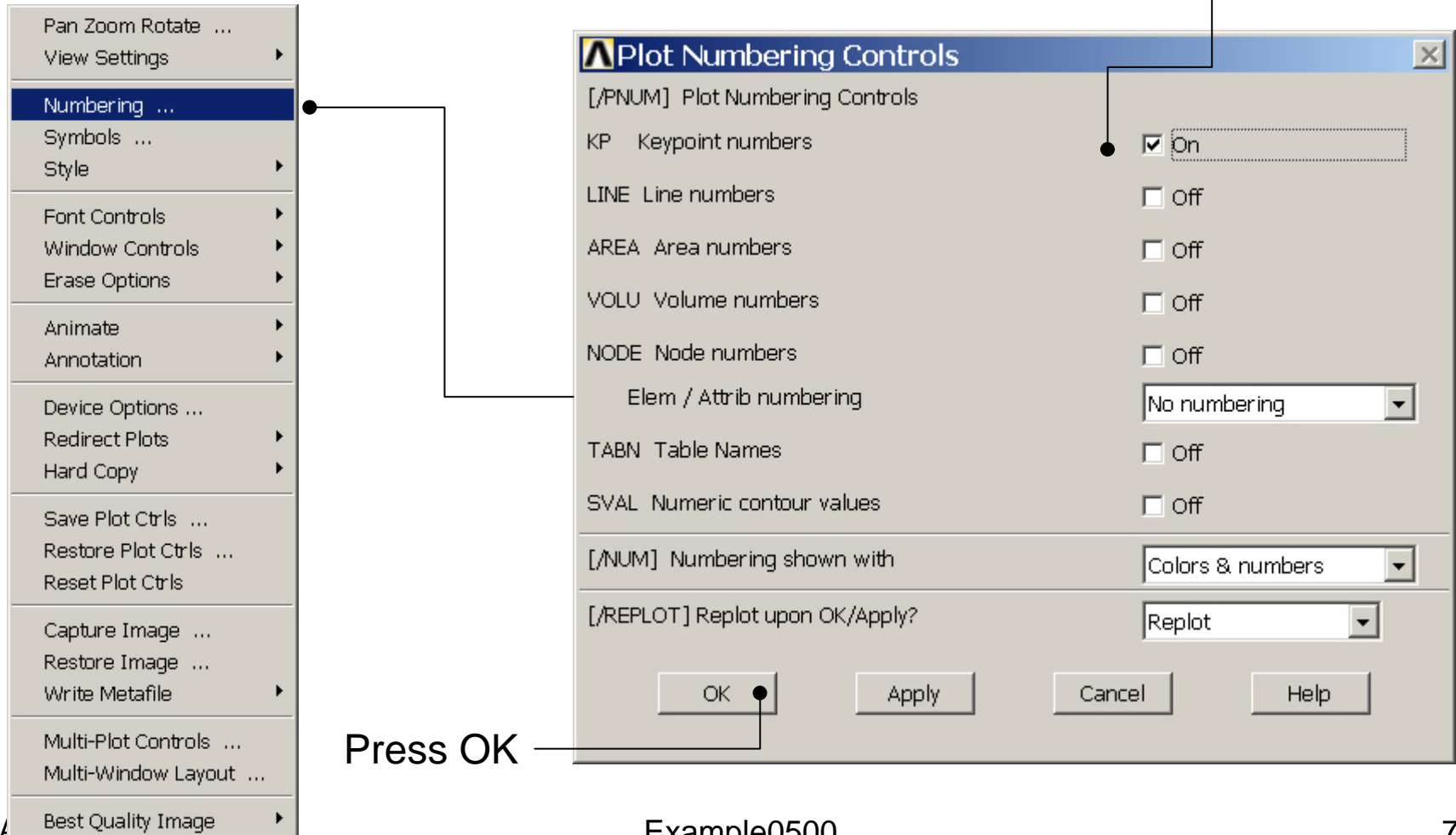


Note: An empty box result in a zero. It is allowed to enter 0.0 in each box.

# Example - Numbering

Utility Menu > PlotCtrls > Numbering

Switch on Keypoint numbers

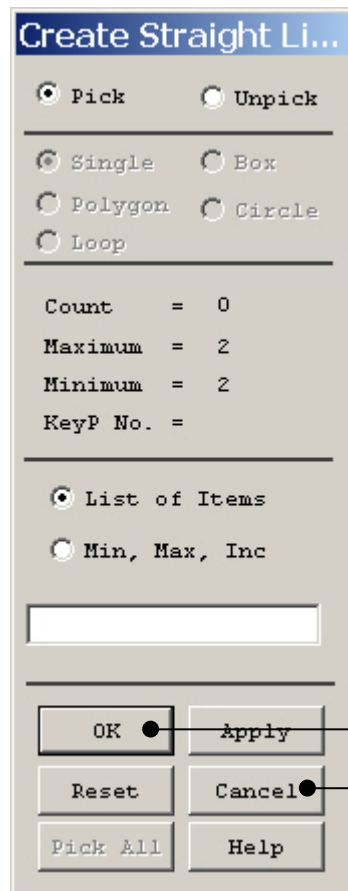


# Example - Lines

**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

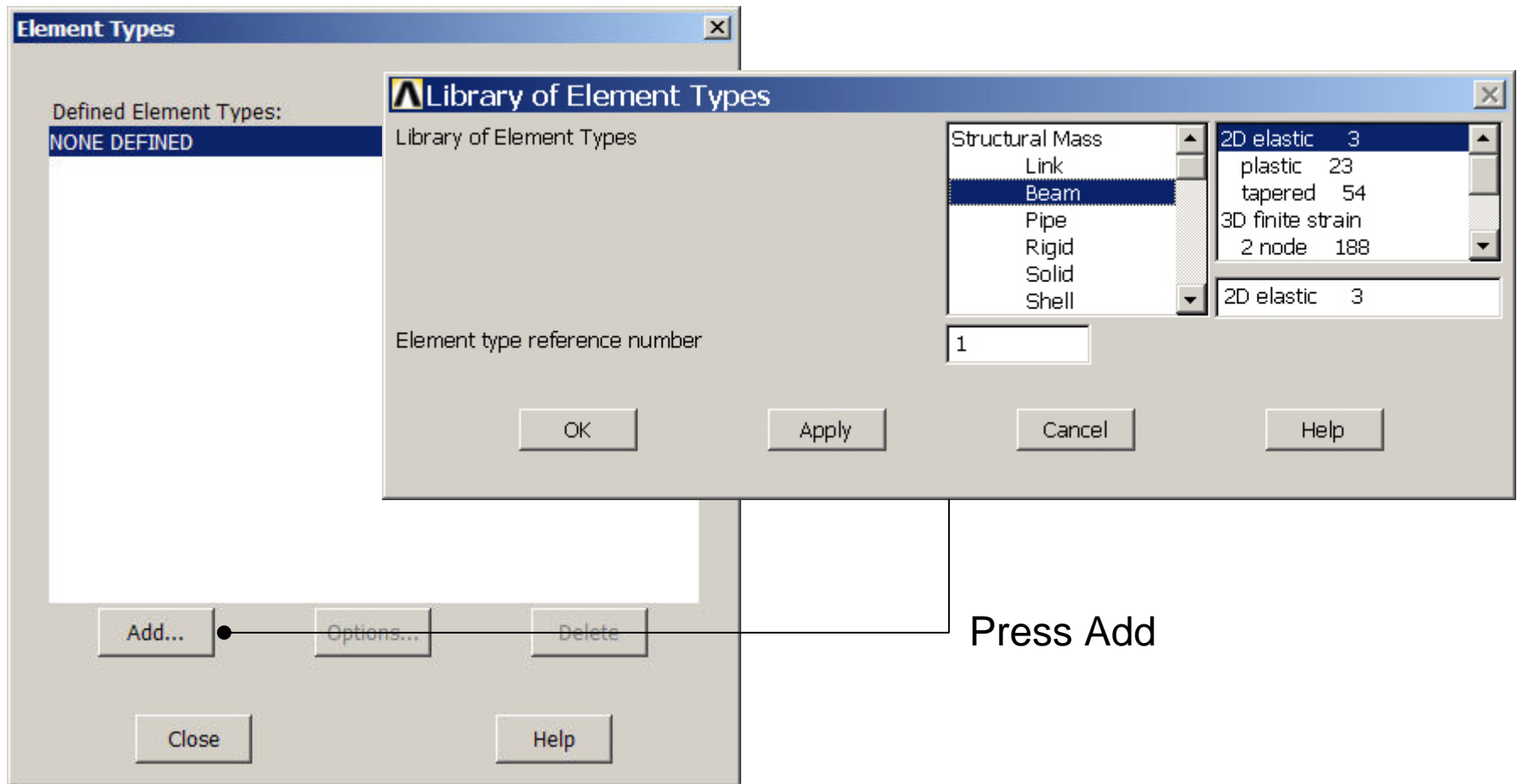
Unpick: downward arrow

Press OK or Cancel to finish selection



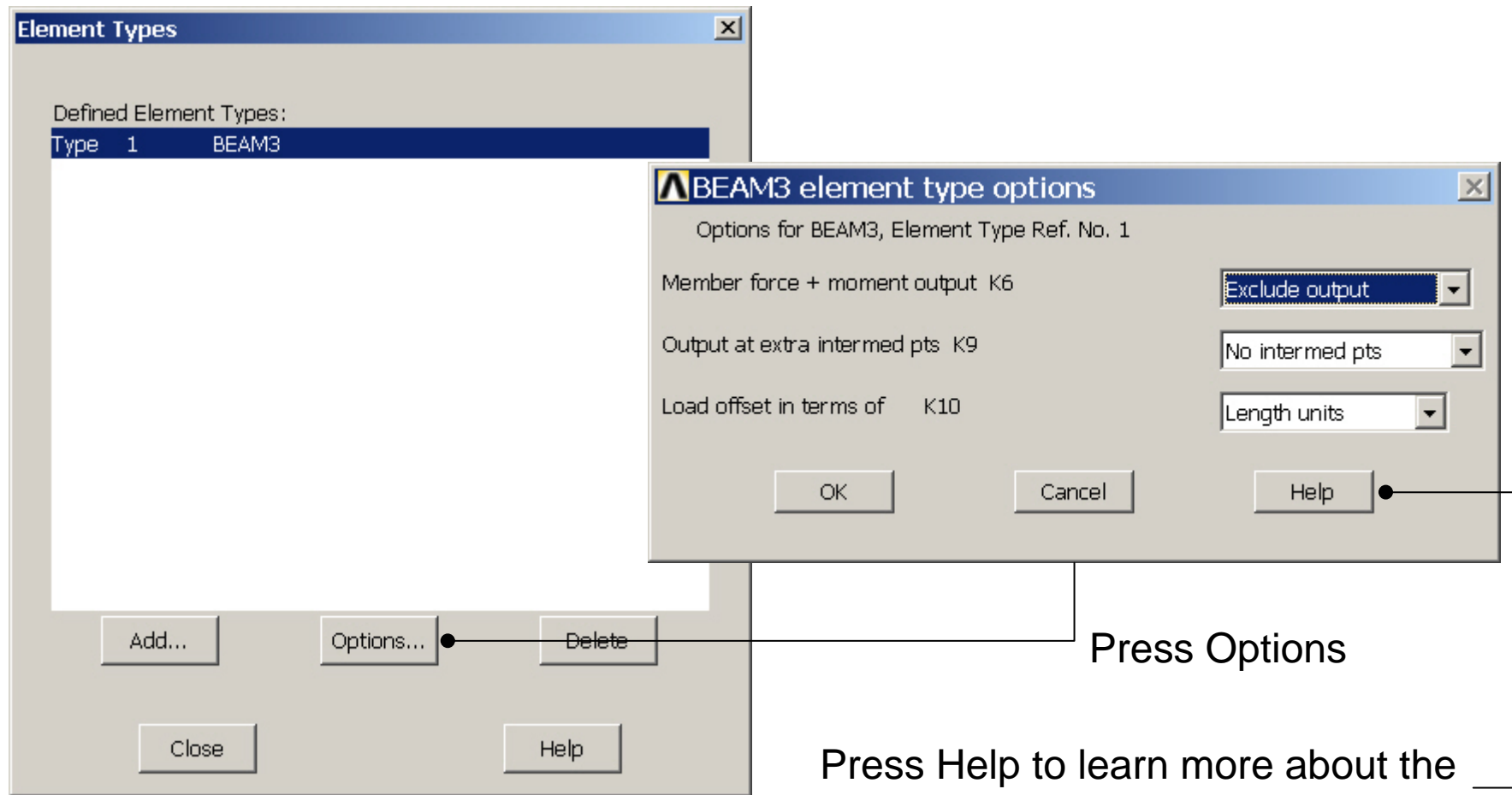
# Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



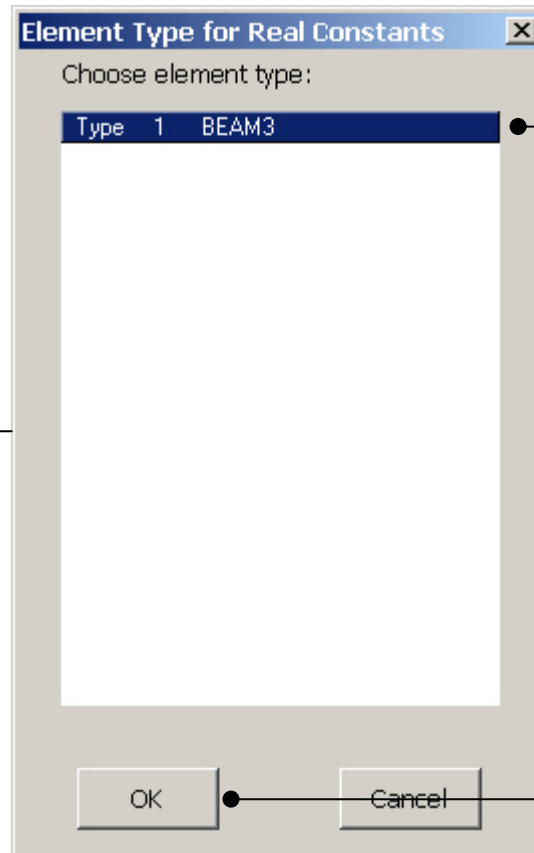
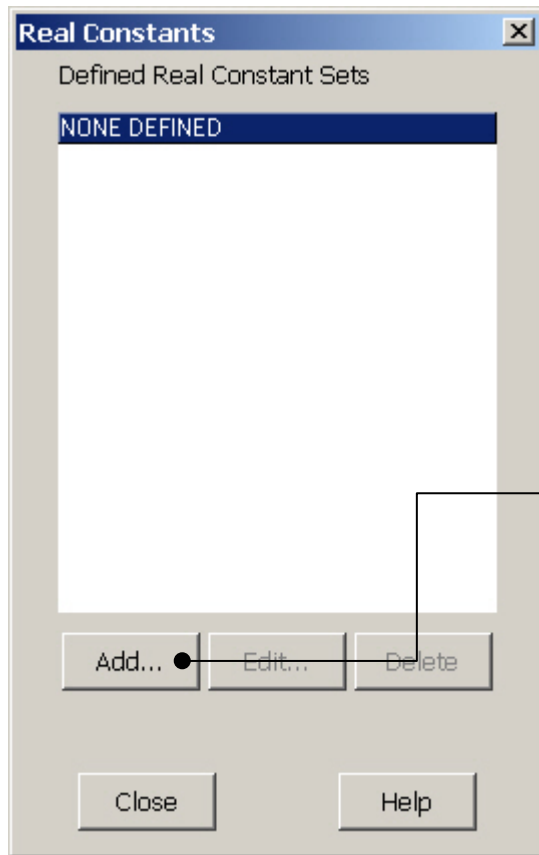
# Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



# Example – Real Constants

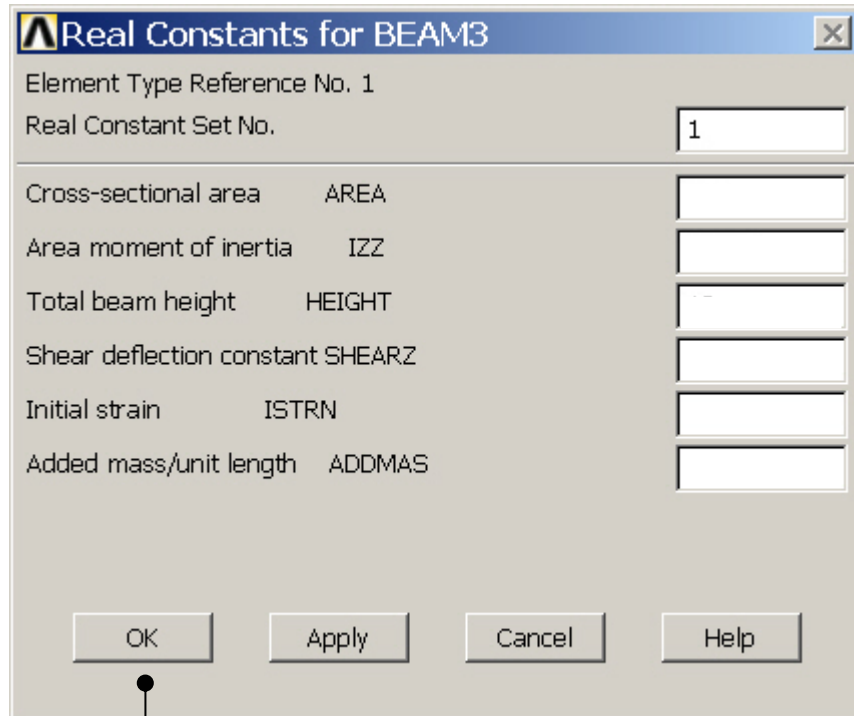
Preprocessor > Real Constants > Add



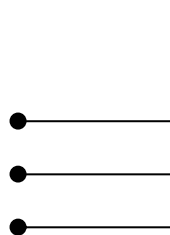
Place the cursor on the relevant element and press OK

# Example - Real Constants

Preprocessor > Real Constants > Add

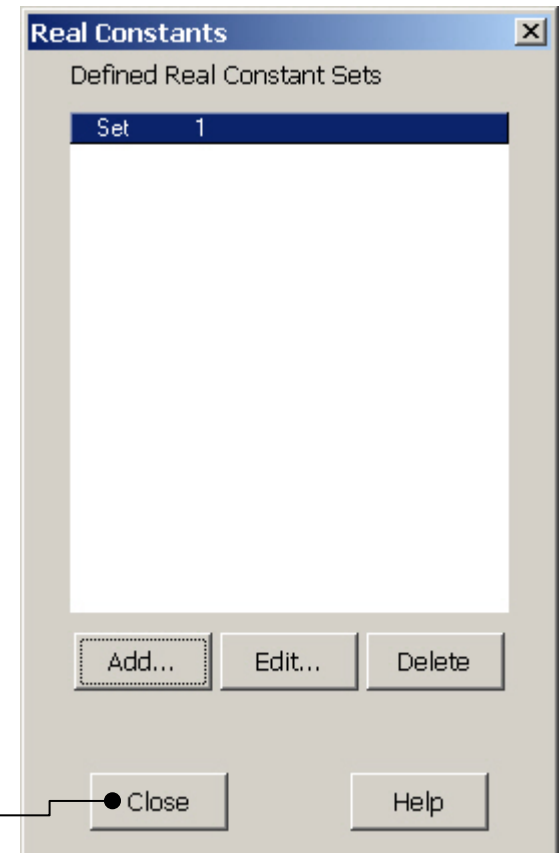


Enter cross-sectional data



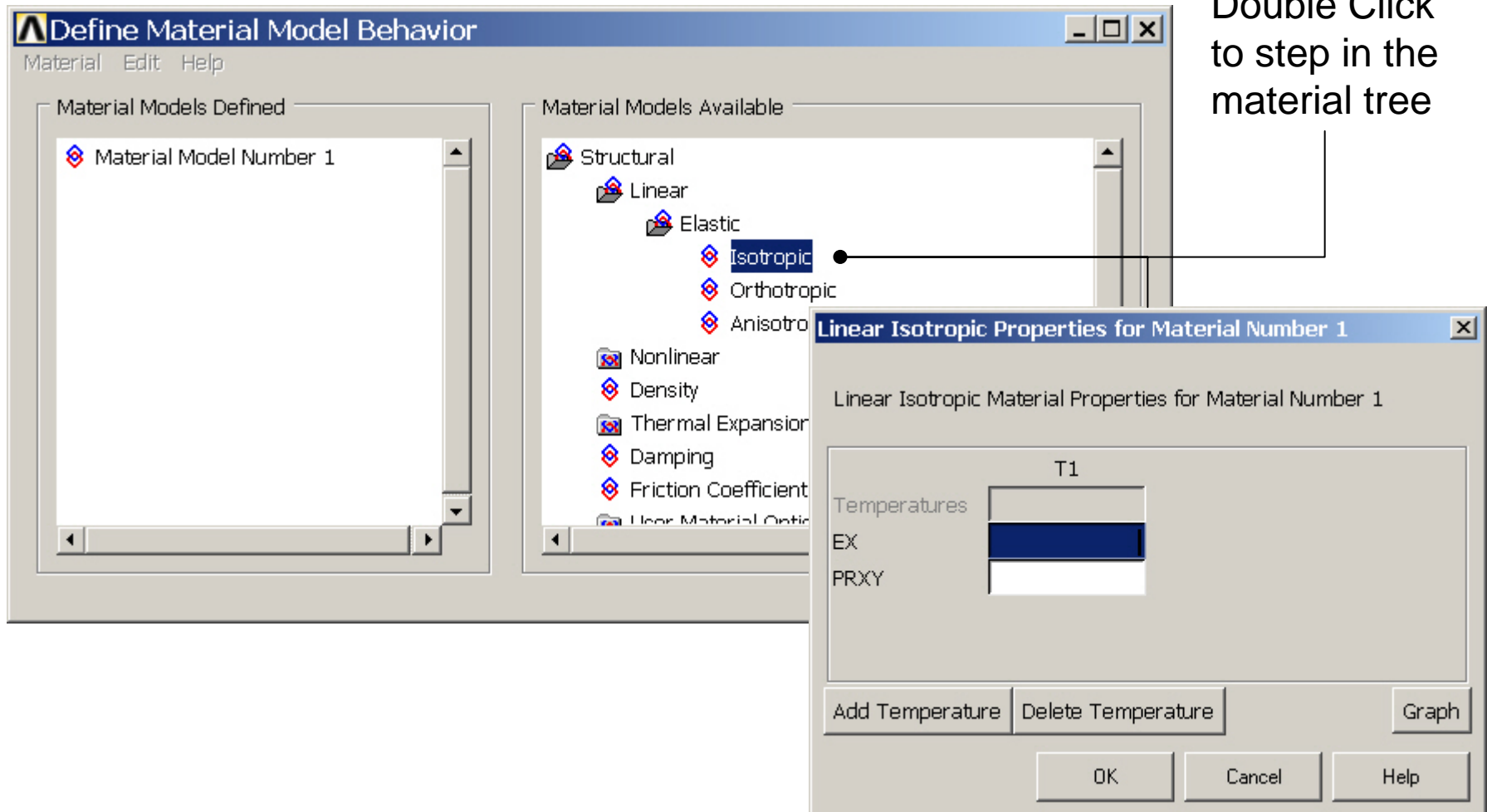
Press OK

Press Close to finish



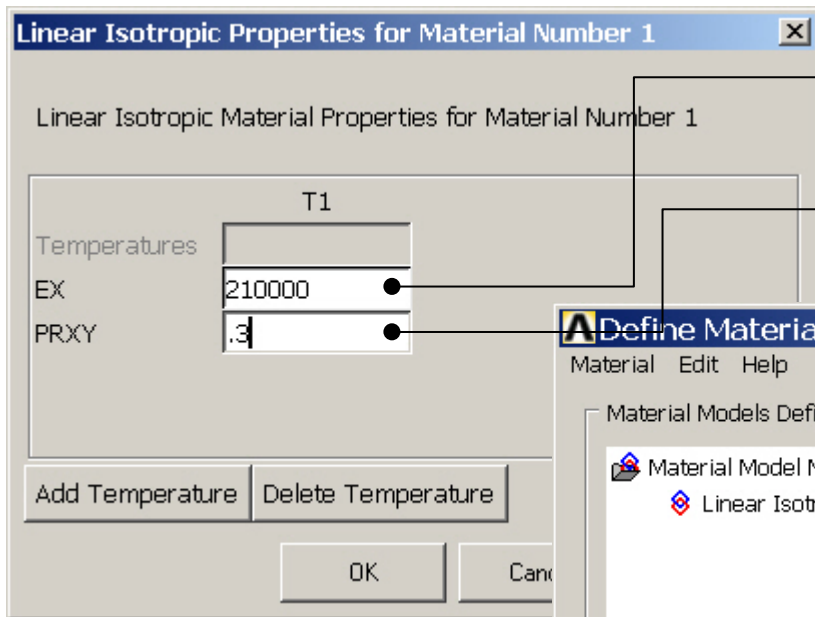
# Example - Material Properties

Preprocessor > Material Props > Material Models



# Example - Material Properties

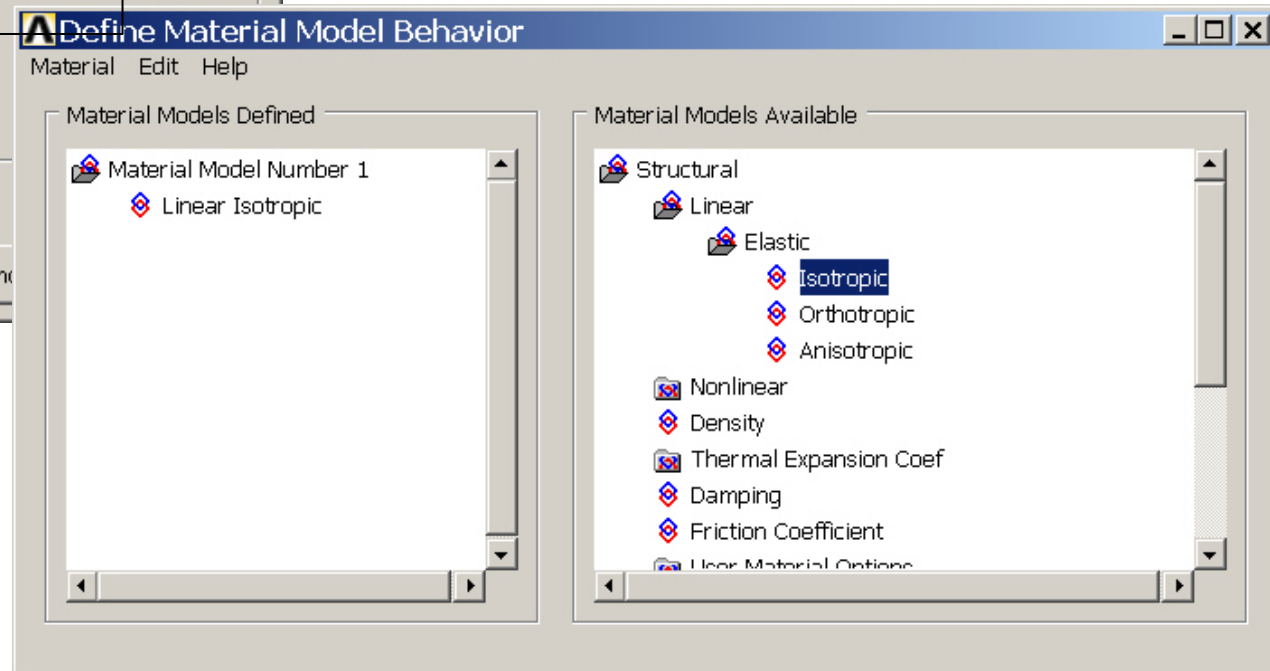
Preprocessor > Material Props > Material Models



Enter:  
Modulus of elasticity

Enter:  
Poisson's ratio

Click here  
to Close



# 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

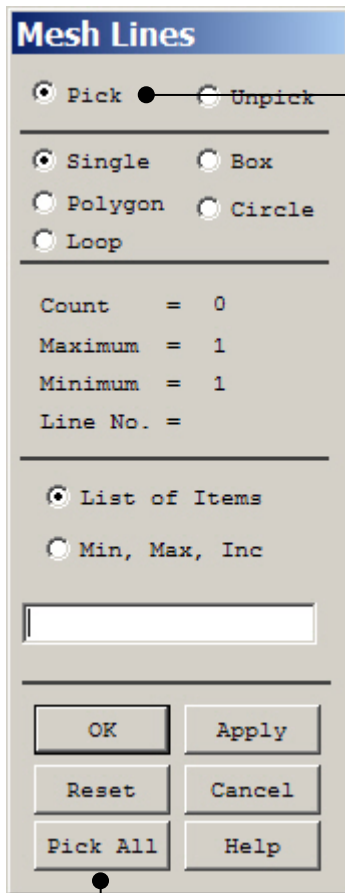
OK Apply Cancel Help

Press OK when finish with selection

Enter 3

# Example - Meshing

Preprocessor > Meshing > Mesh > Lines



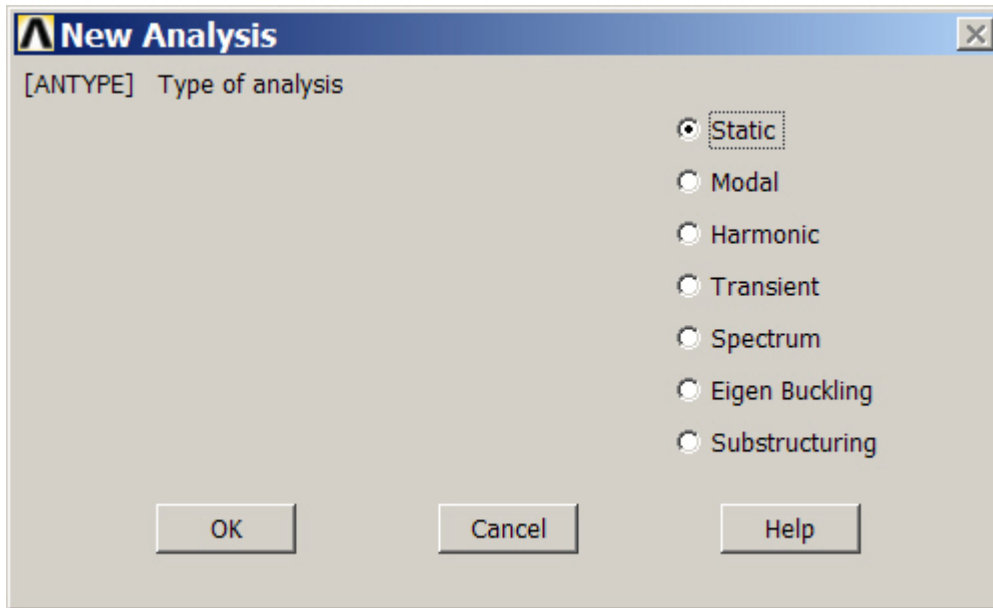
Select individual lines to be meshed

Select all lines defined to be meshed



# Example – Analysis Type

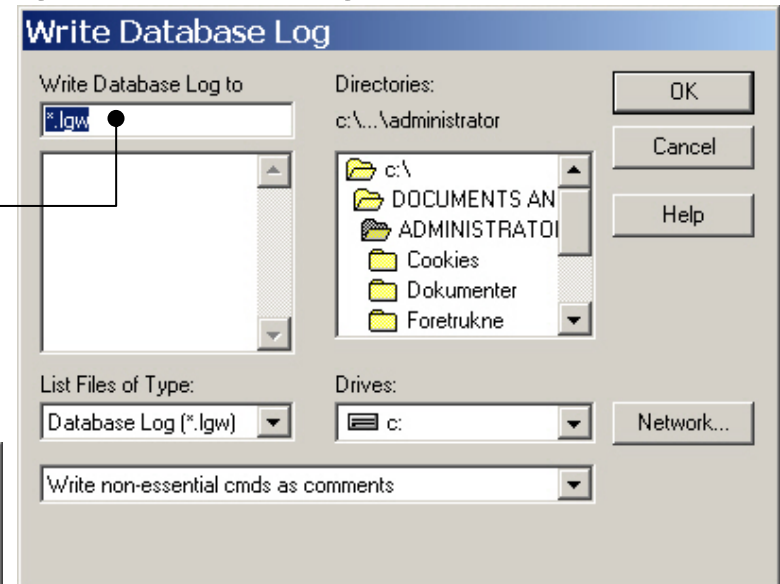
**Solution > Analysis Type > New Analysis**



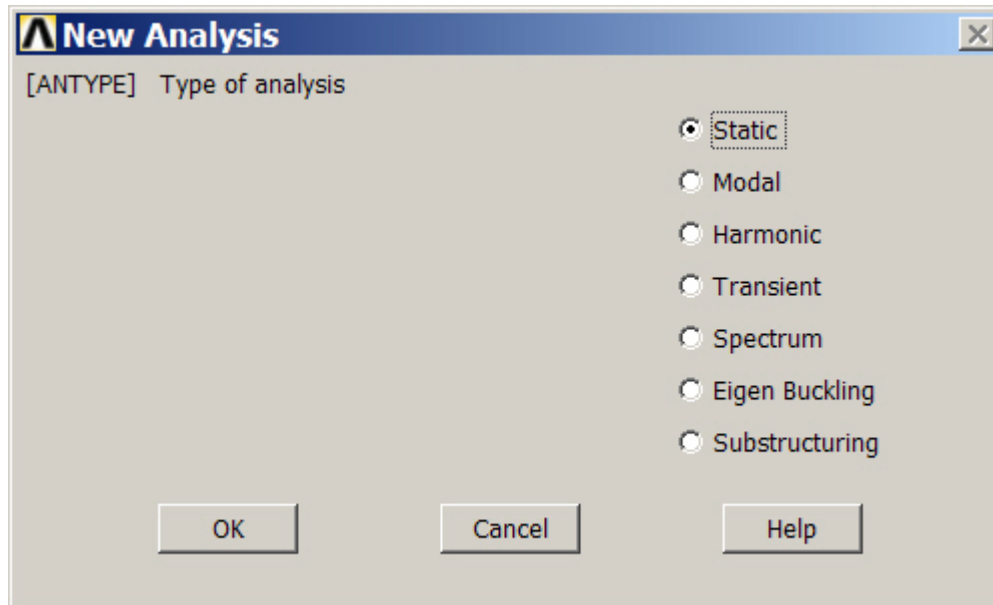
# Example – Analysis Type

**File > Write DB log file**

Enter “example0500.lgw”



**Solution > Analysis Type > New Analysis**



# 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**

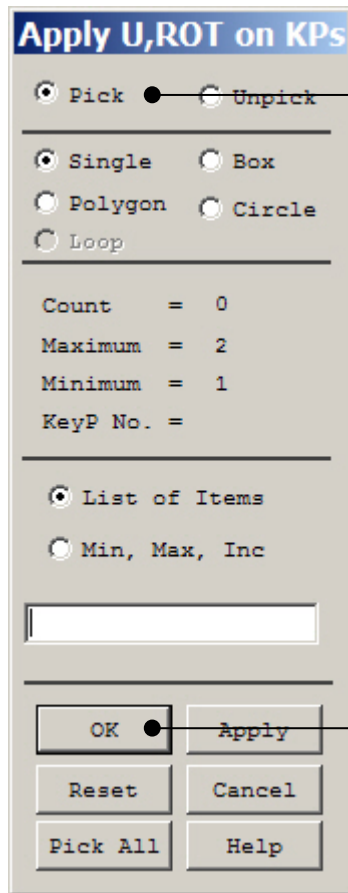
**Activate the Unabridged menu**

OK Cancel Help

Example0500

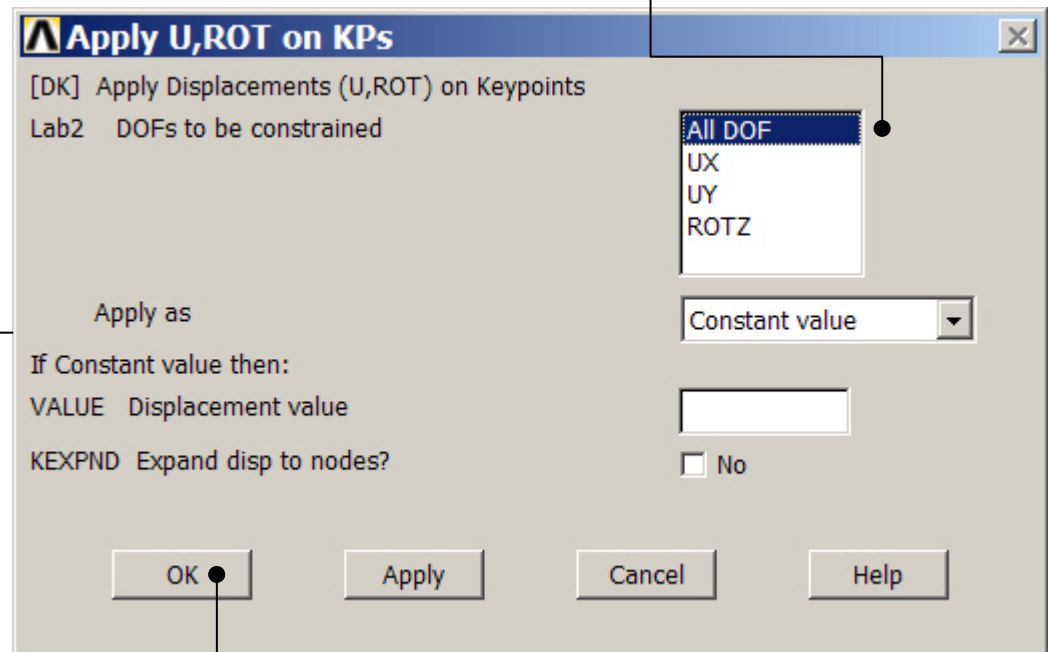
# Example – Define Loads

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



Select keypoint 1

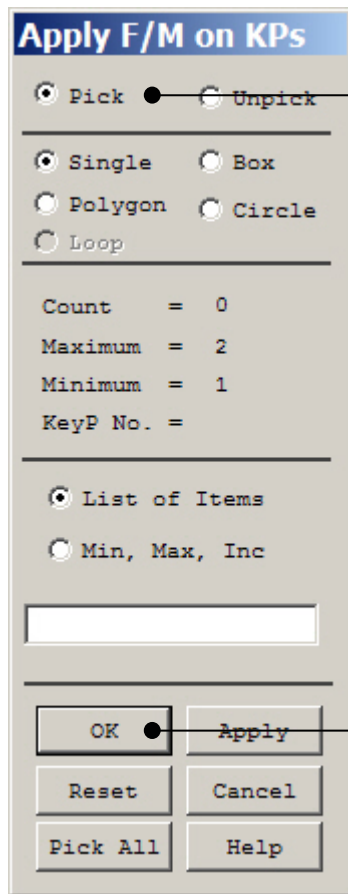
Select All DOF to fix/clamp the beam



Press OK

# Example – Define Loads

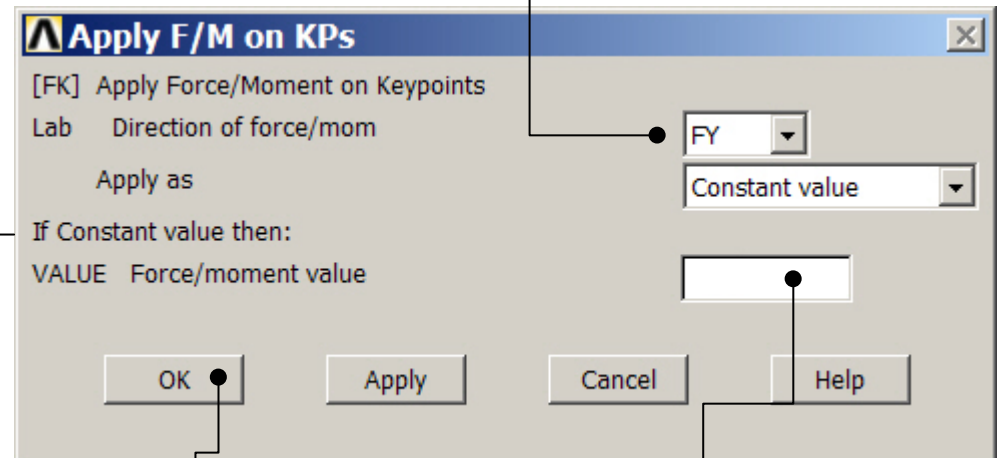
**Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints**



Select keypoint 2

Note: As a unit load is specified, the load factors represent the buckling loads

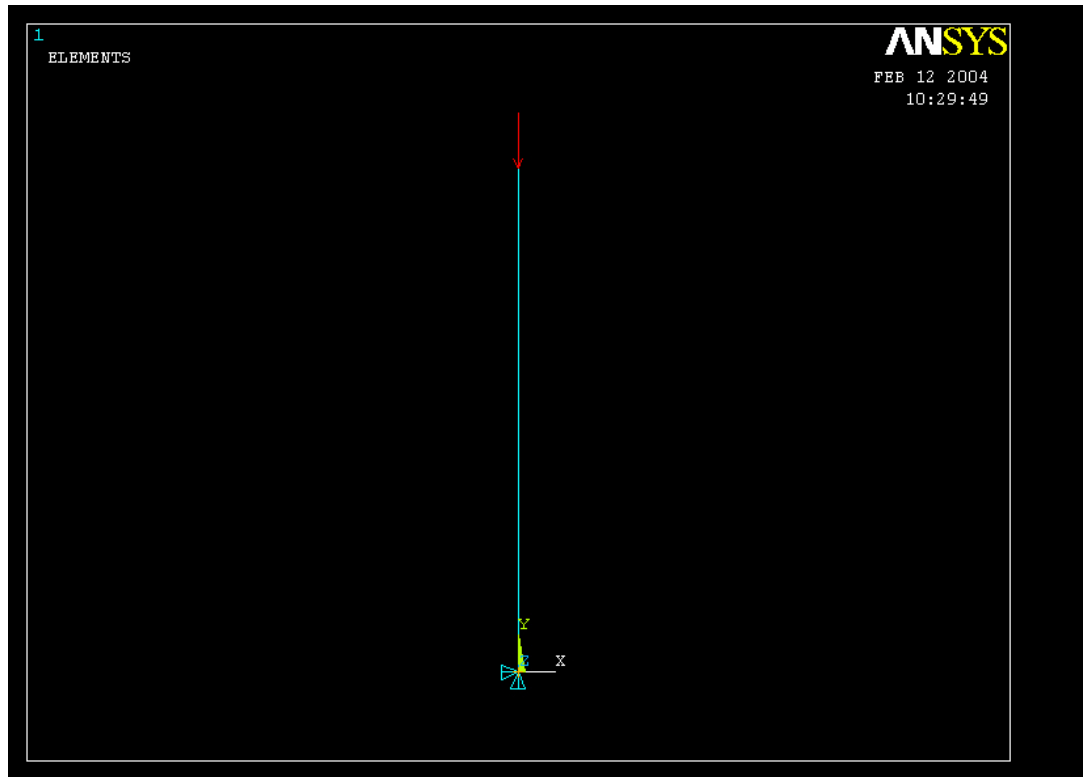
Change to FY



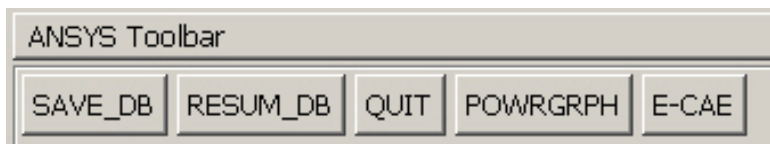
Press OK to finish

Enter -1

# Example - Save



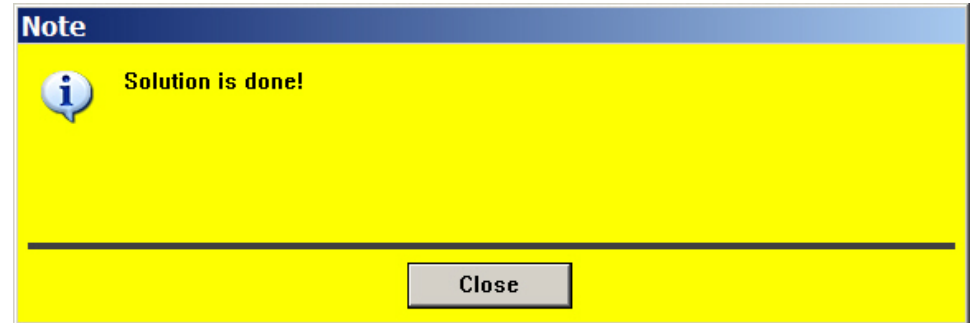
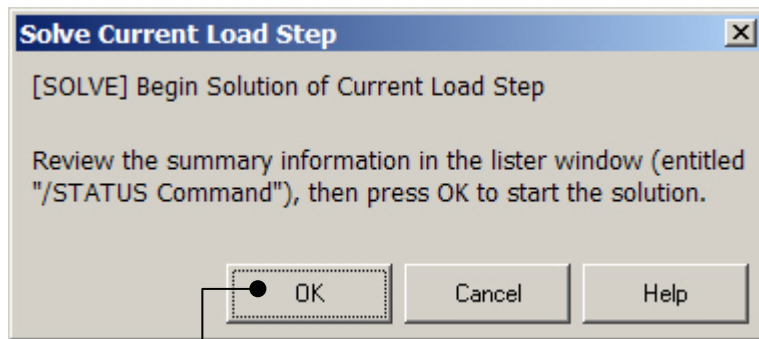
Display of Analysis model



Save the model

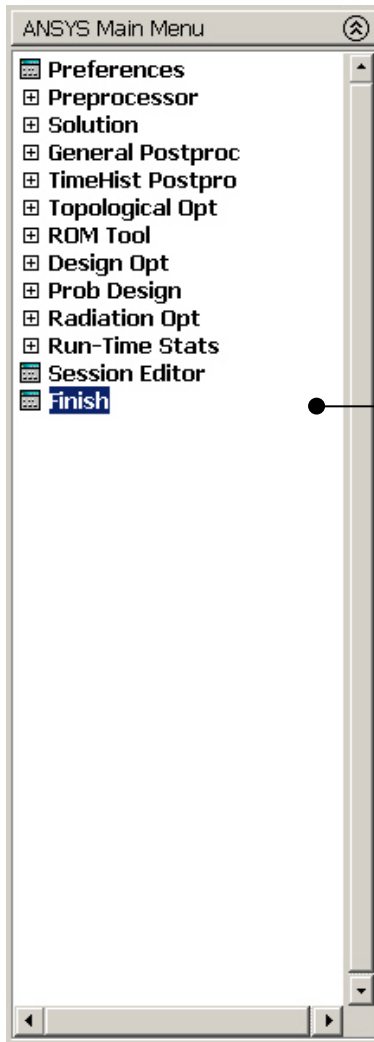
# Example - Solve

**Solution > Solve > Current LS**



Press OK

# Example - Finish

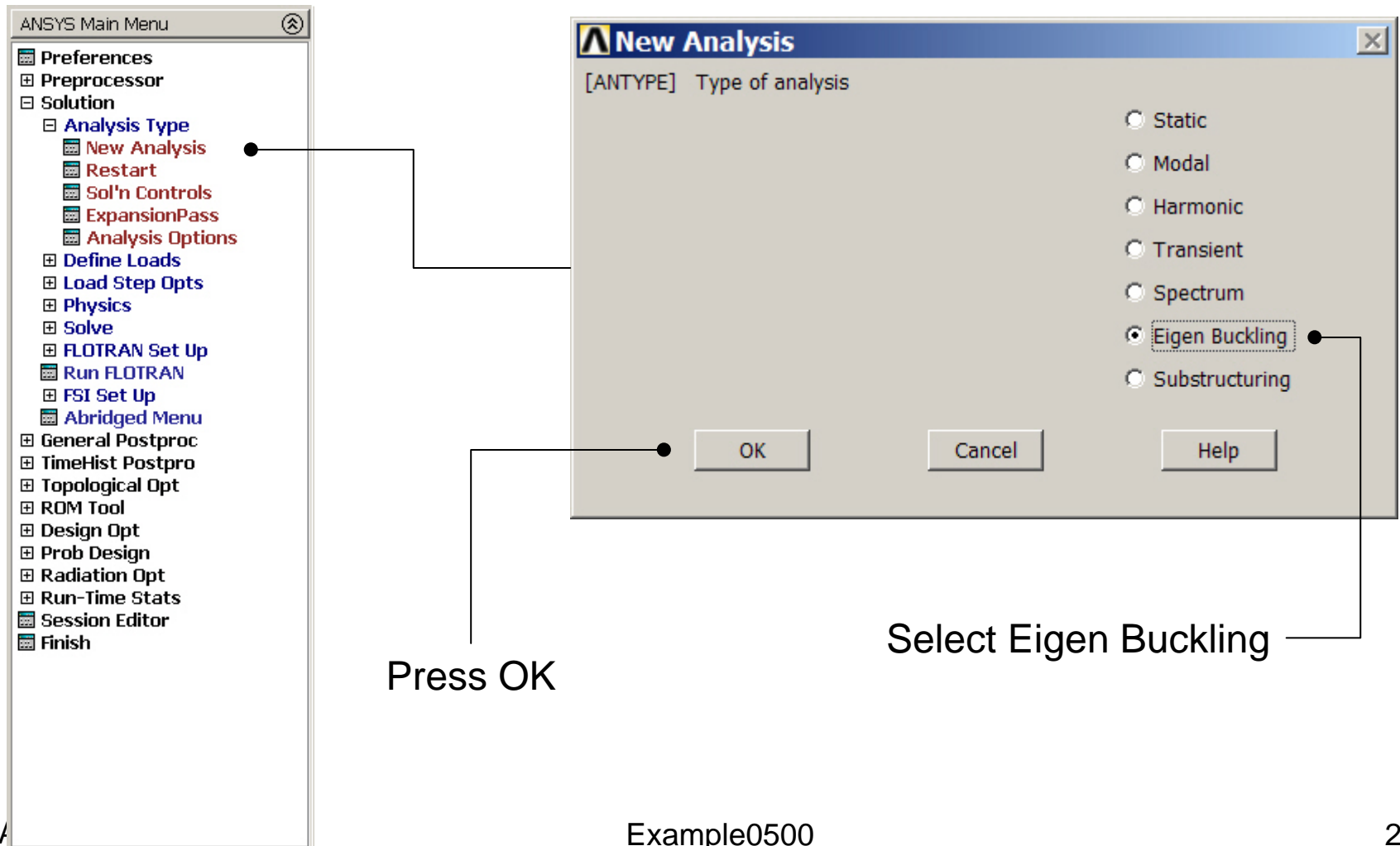


Press Finish to end the static solution



# Eigen Buckling - New Analysis

Main Menu> Solution> Analysis Type> New Analysis

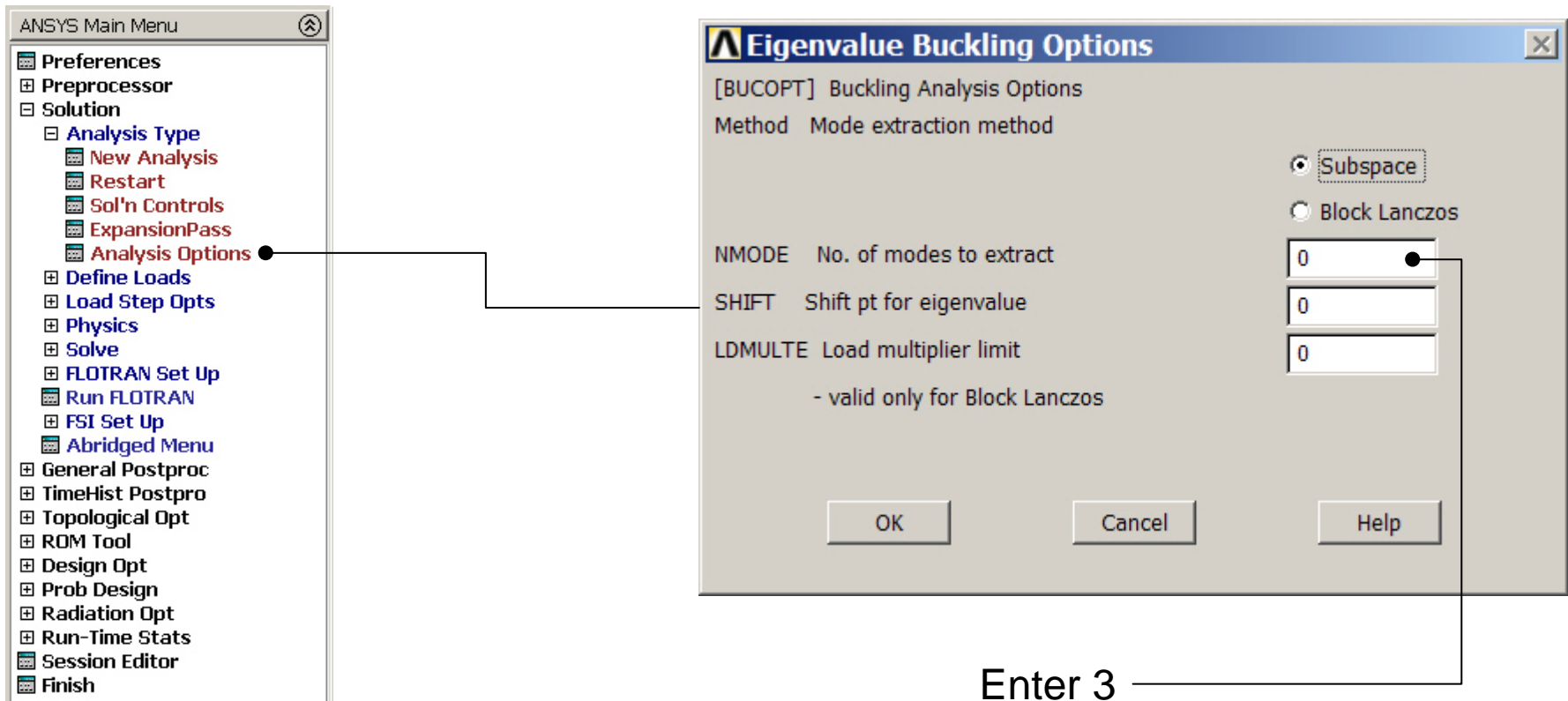


Example0500

25

# Eigen Buckling – Analysis Options

Main Menu> Solution> Analysis Type> Analysis Options



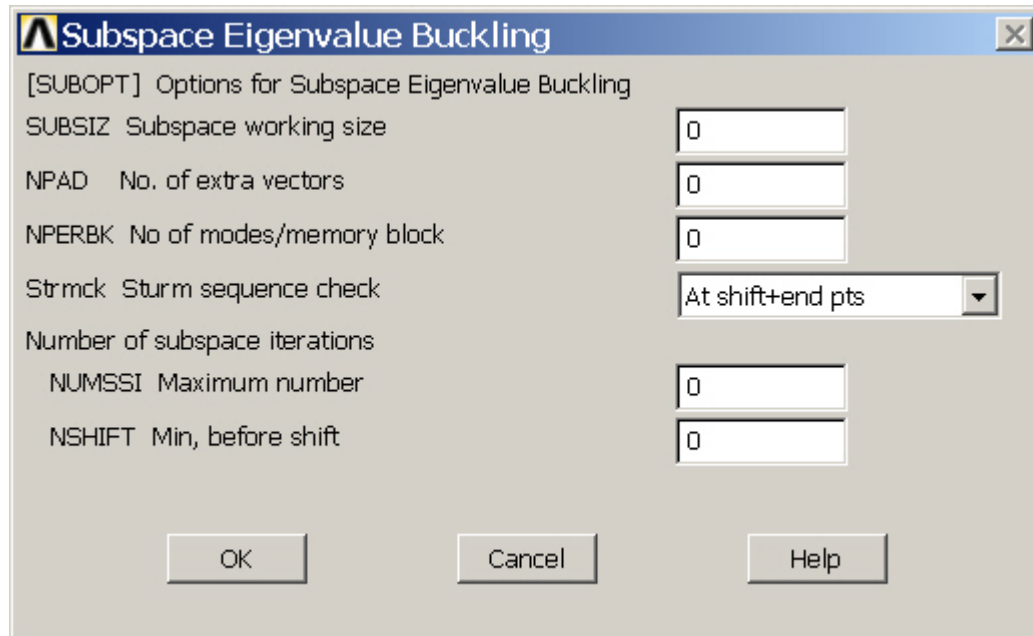
Example0500

26

# 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.

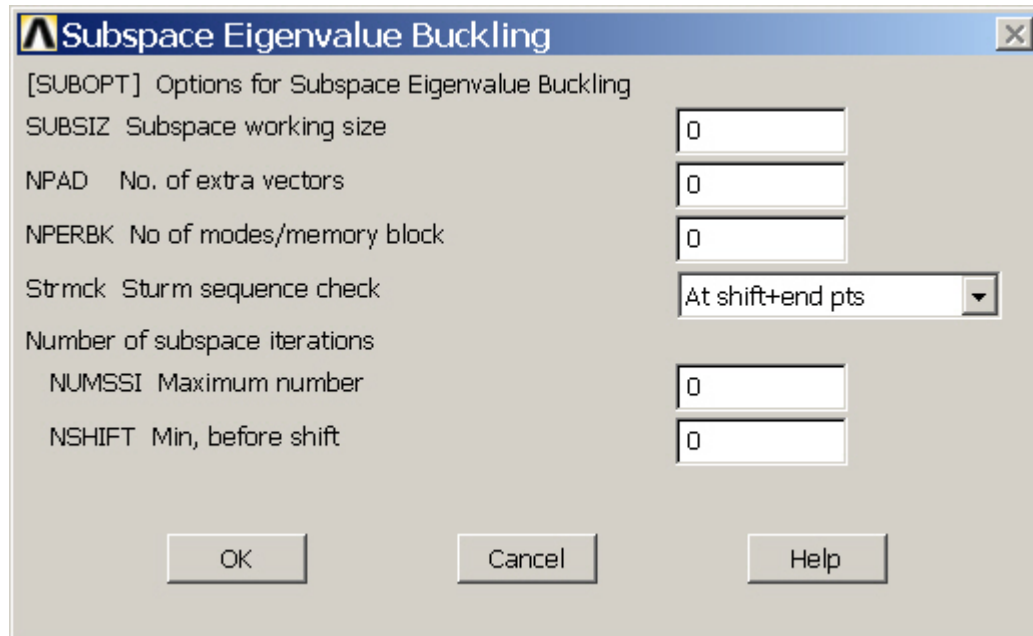
# Example – Subspace Options



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).

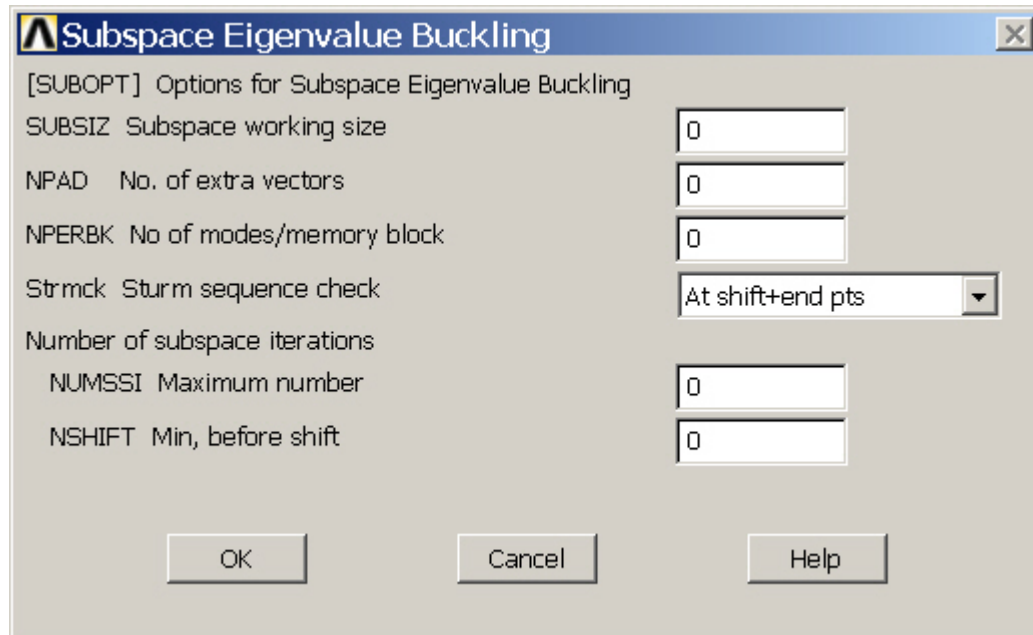
Number of extra vectors used in the iterations. Defaults to 4. The total number of vectors used is  $NMODE + NPAD$ .

# 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.

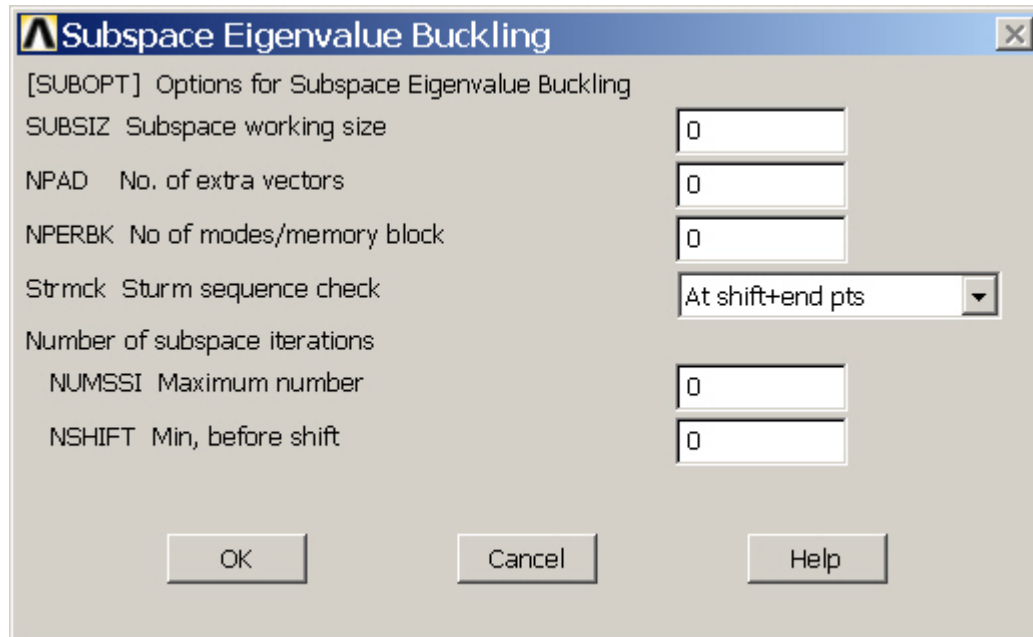
# Example – Subspace Options



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 --

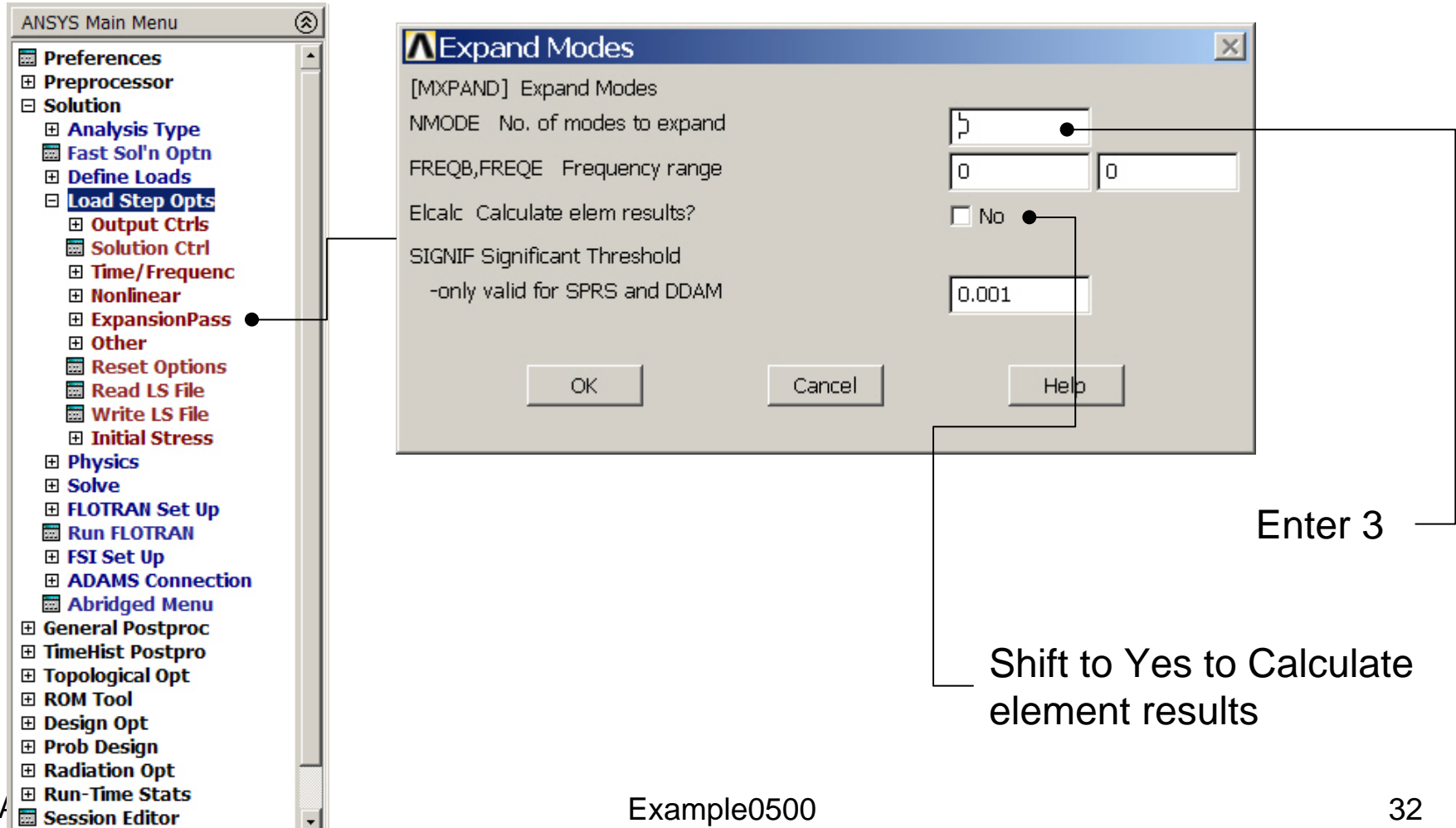
Perform check only at all shift points.

NONE --

Do not perform Sturm sequence check.

# Eigen Buckling – Expanding Modes

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

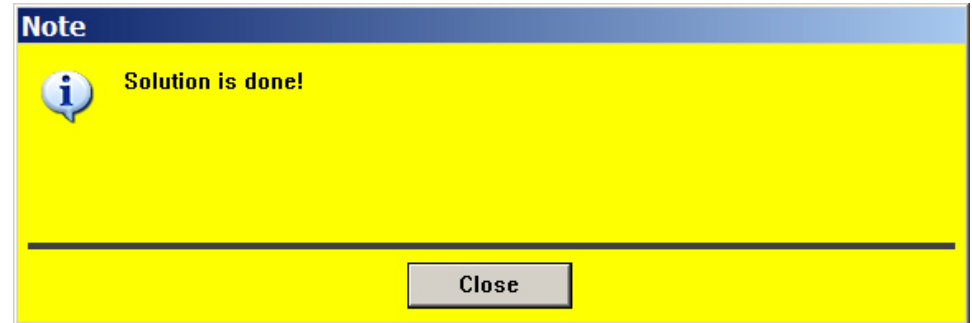
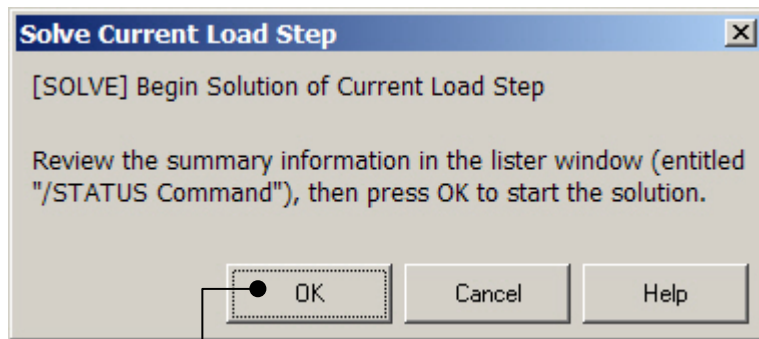


Example0500



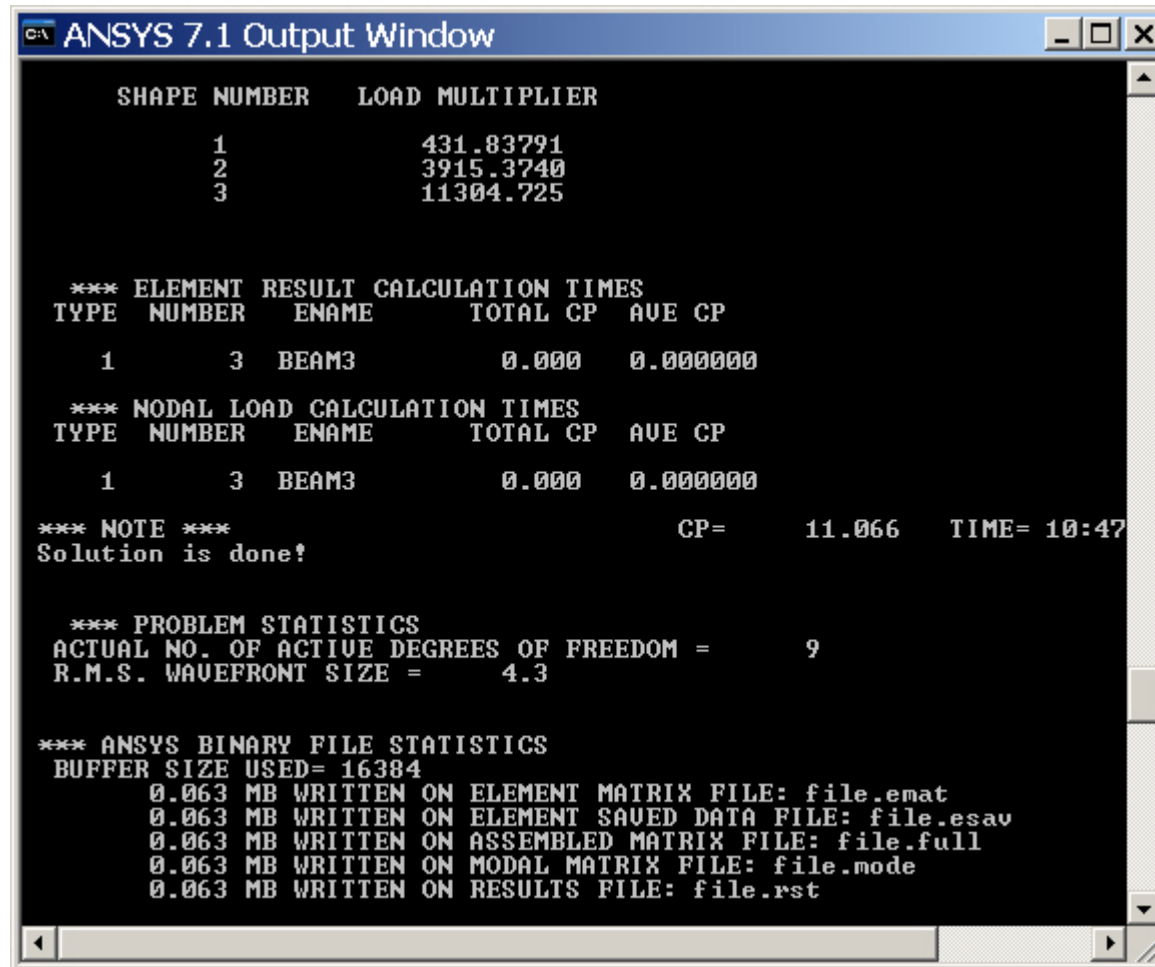
# Example - Solve

**Solution > Solve > Current LS**



Press OK

# Example – Output Window



The screenshot shows the ANSYS 7.1 Output Window with a black background and white text. The window title is 'C:\ ANSYS 7.1 Output Window'. The output text is as follows:

```
SHAPE NUMBER      LOAD MULTIPLIER
      1             431.83791
      2             3915.3740
      3             11304.725

*** ELEMENT RESULT CALCULATION TIMES
TYPE  NUMBER  ENAME      TOTAL CP  AVE CP
   1      3  BEAM3         0.000   0.000000

*** NODAL LOAD CALCULATION TIMES
TYPE  NUMBER  ENAME      TOTAL CP  AVE CP
   1      3  BEAM3         0.000   0.000000

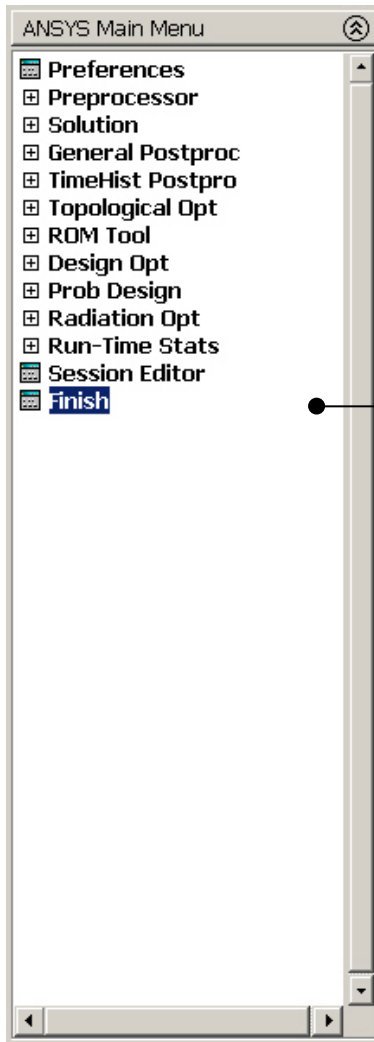
*** NOTE ***
Solution is done!

*** PROBLEM STATISTICS
ACTUAL NO. OF ACTIVE DEGREES OF FREEDOM =      9
R.M.S. WAVEFRONT SIZE =      4.3

*** ANSYS BINARY FILE STATISTICS
BUFFER SIZE USED= 16384
0.063 MB WRITTEN ON ELEMENT MATRIX FILE: file.emat
0.063 MB WRITTEN ON ELEMENT SAVED DATA FILE: file.esav
0.063 MB WRITTEN ON ASSEMBLED MATRIX FILE: file.full
0.063 MB WRITTEN ON MODAL MATRIX FILE: file.mode
0.063 MB WRITTEN ON RESULTS FILE: file.rst
```

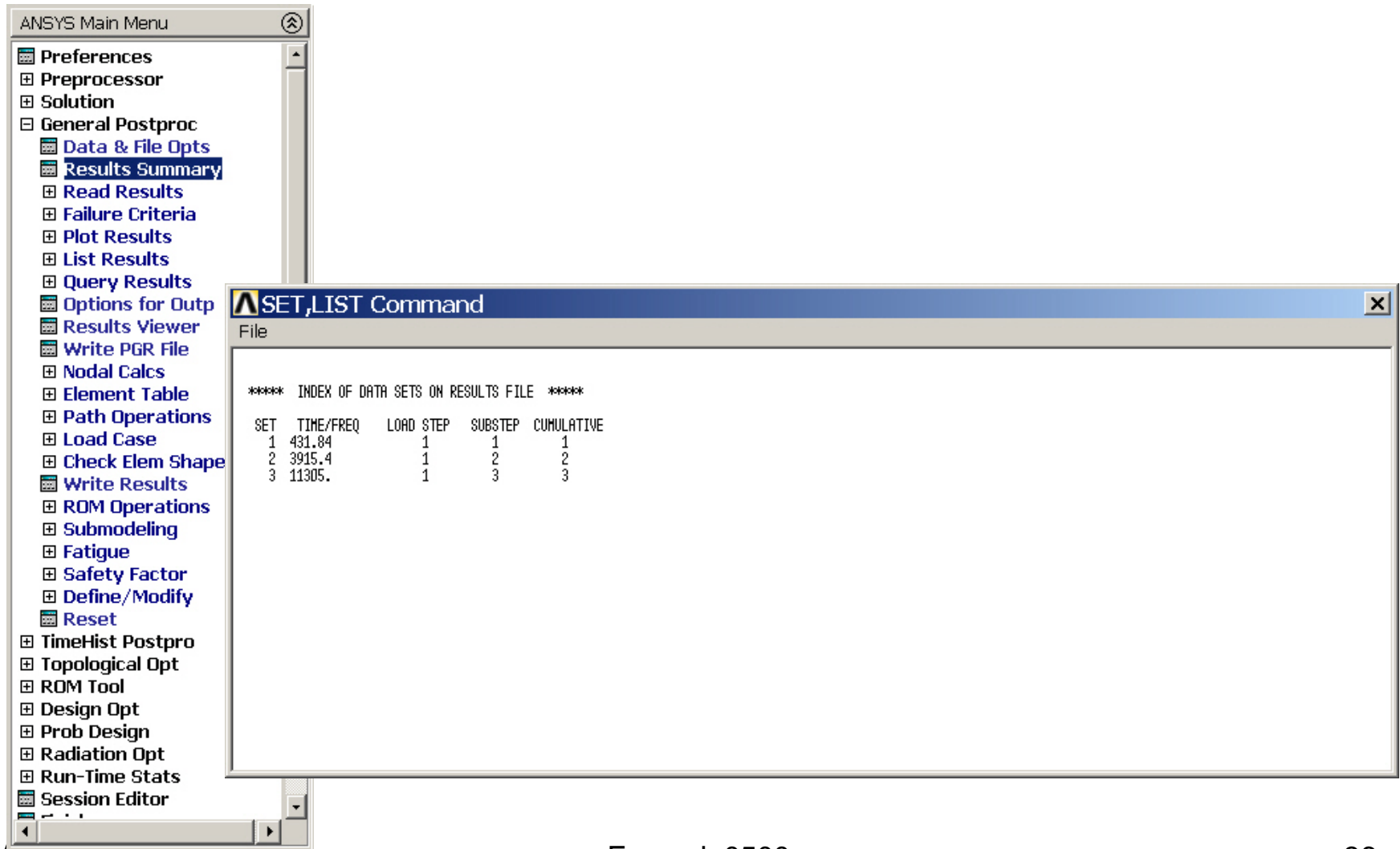
At the bottom right of the window, the text 'CP= 11.066 TIME= 10:47' is visible.

# Example - Finish



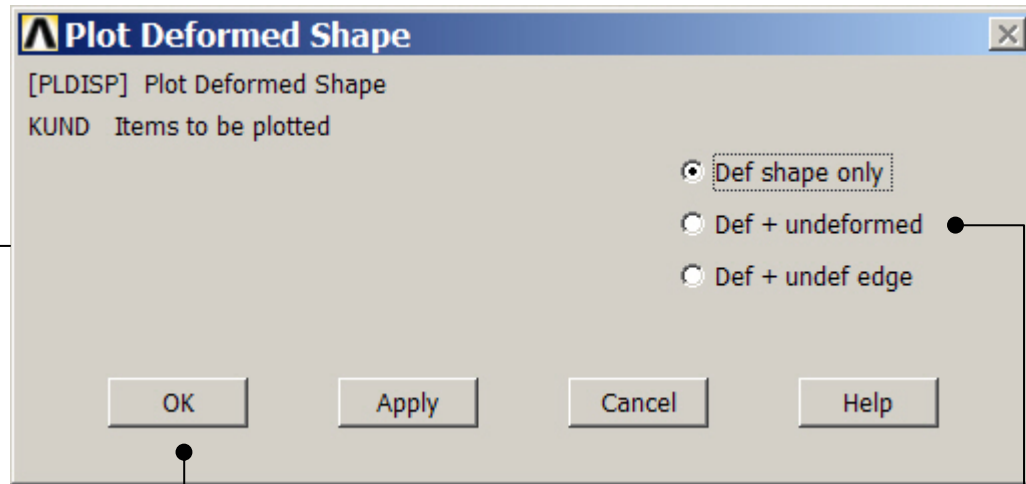
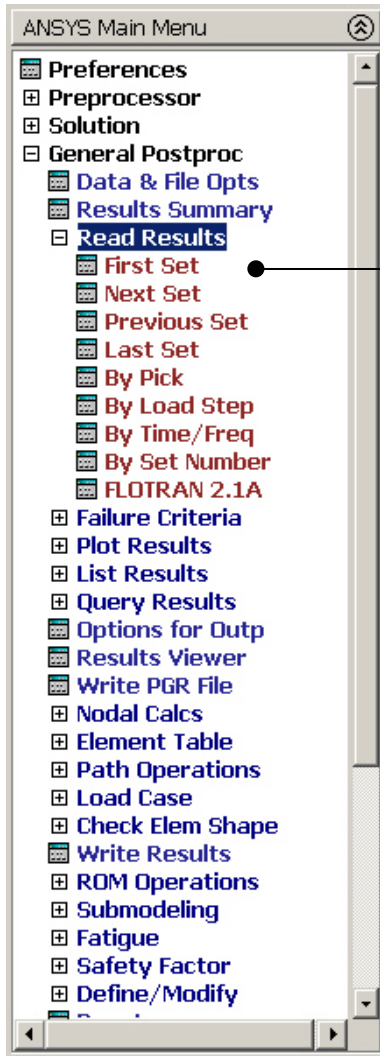
Press Finish to end the eigen buckling solution

# Example – Results Summary



# Example – Read Results

General Postproc > Plot Results > Deformed Shape



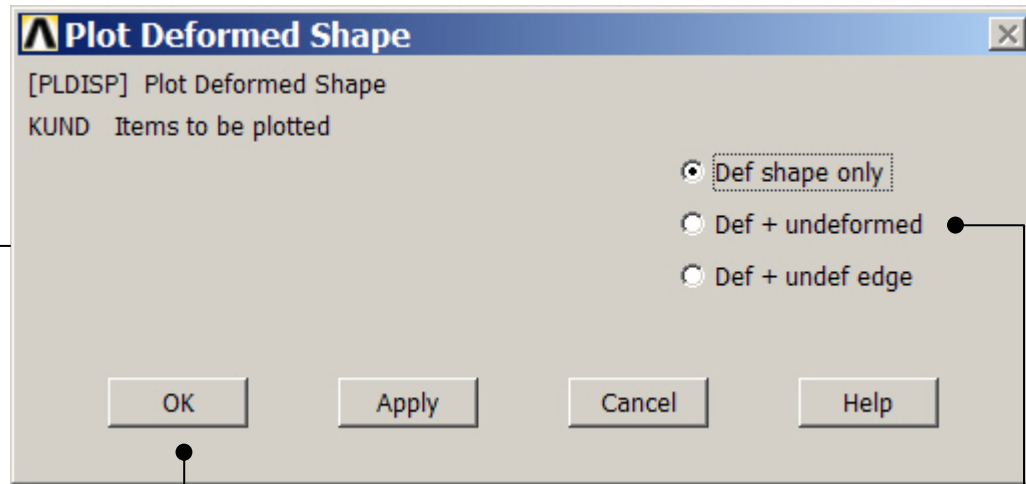
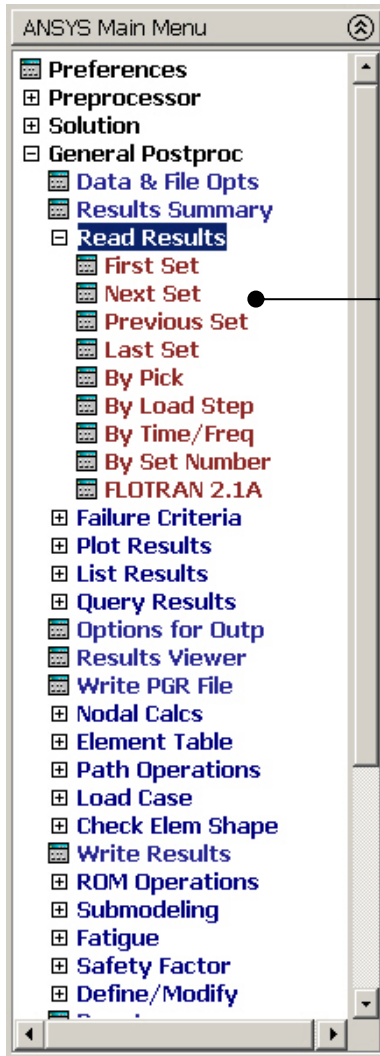
Select "Def+undeformed"  
and Press OK

# Example - PostProcessing



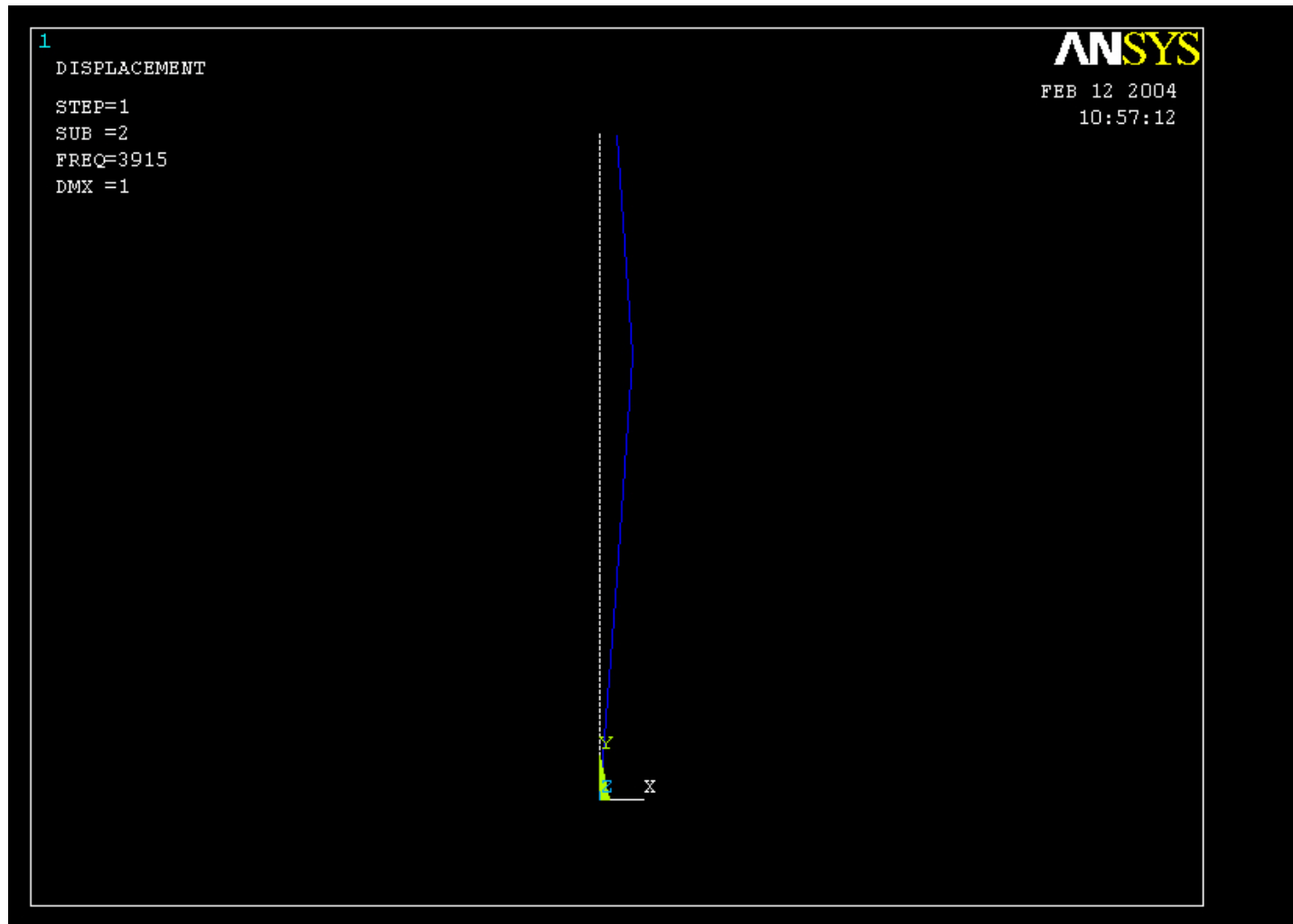
# Example – Read Results

General Postproc > Plot Results > Deformed Shape



Select "Def+undeformed"  
and Press OK

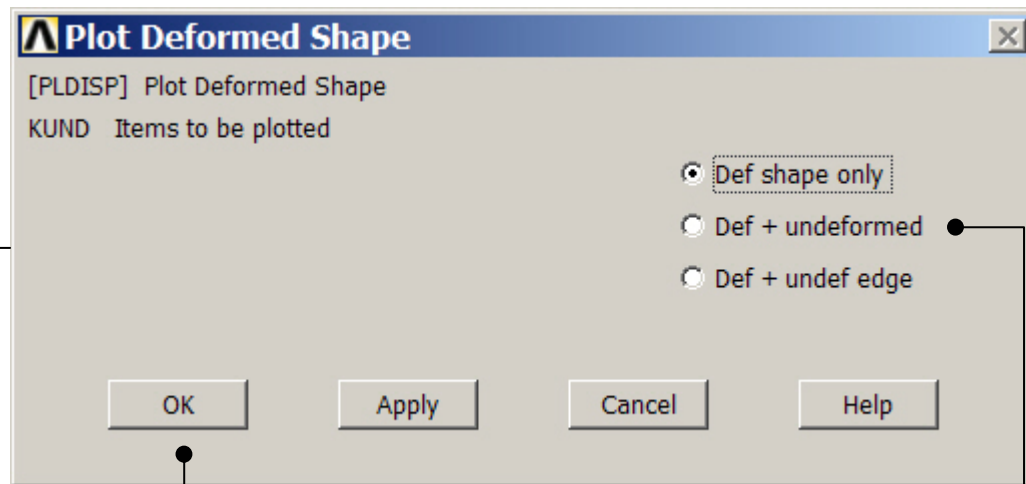
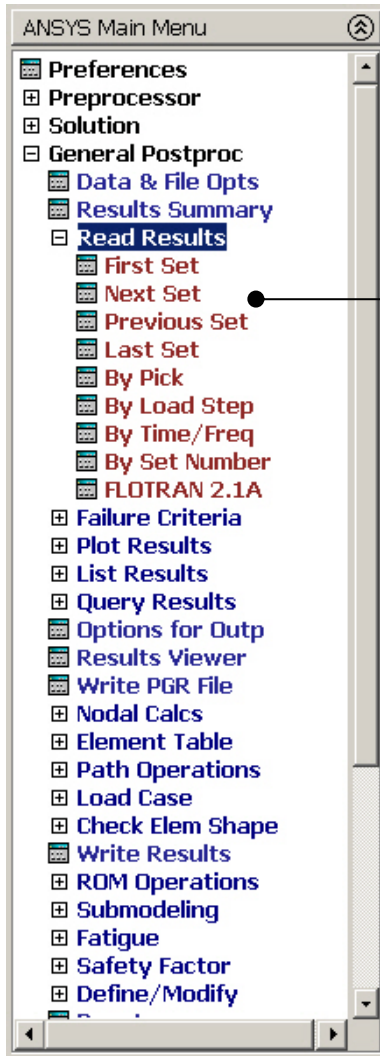
# Example - PostProcessing





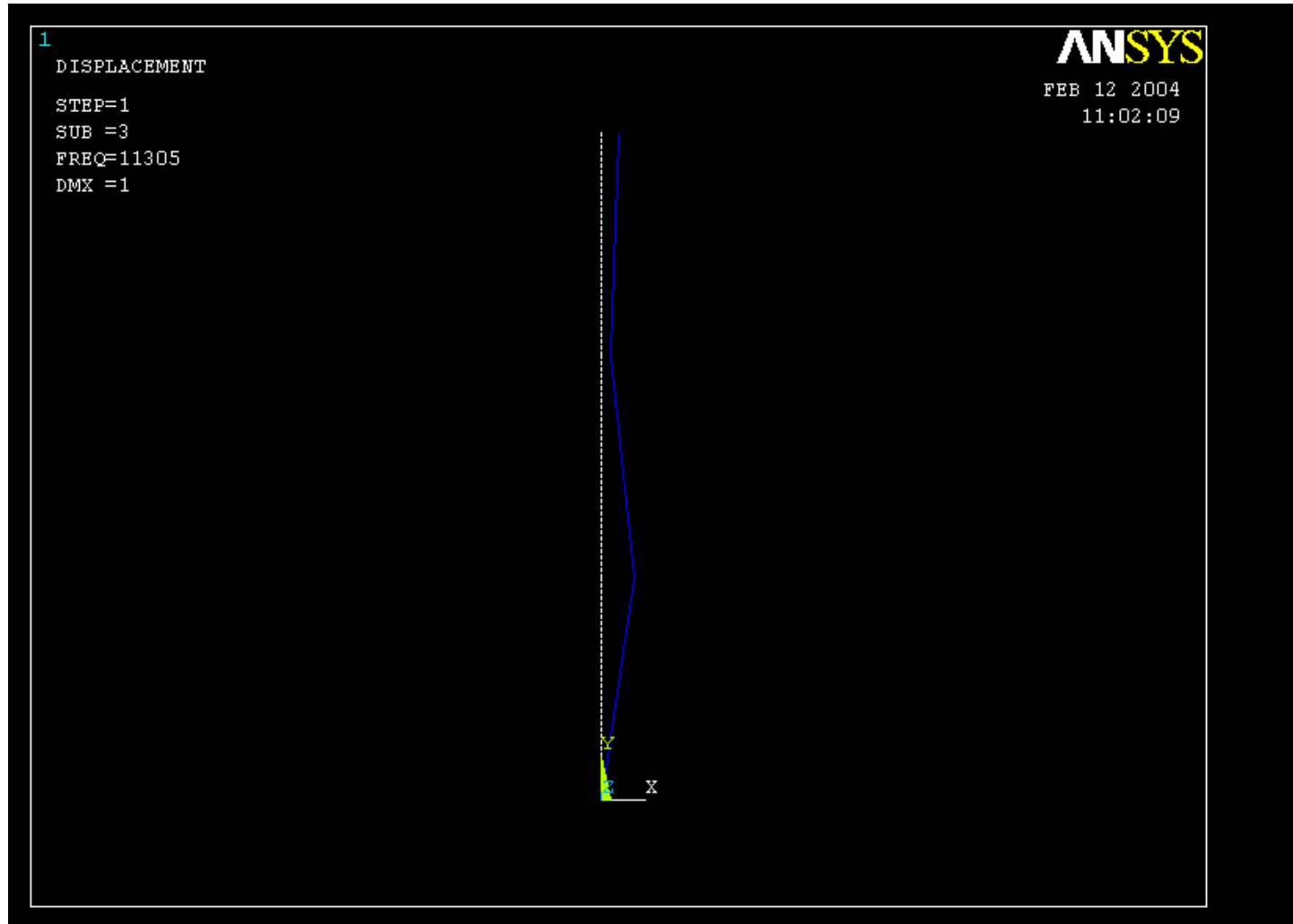
# Example – Read Results

General Postproc > Plot Results > Deformed Shape

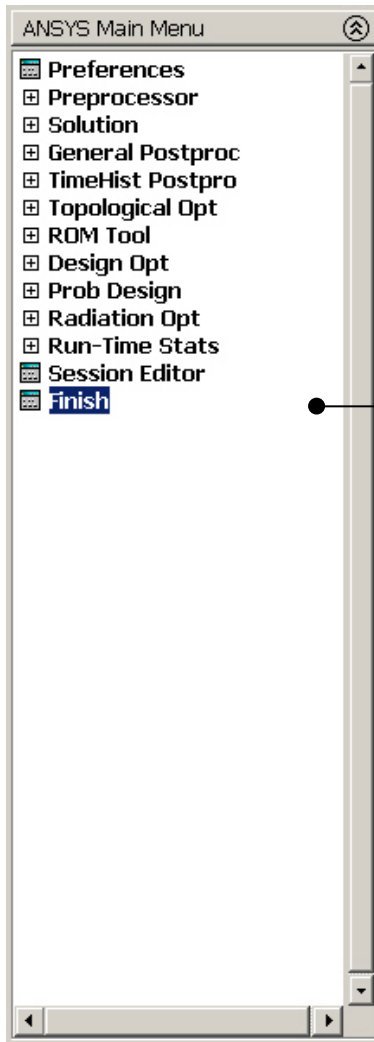


Select "Def+undeformed"  
and Press OK

# Example - PostProcessing



# Example - Finish



Press Finish to end the eigen buckling solution