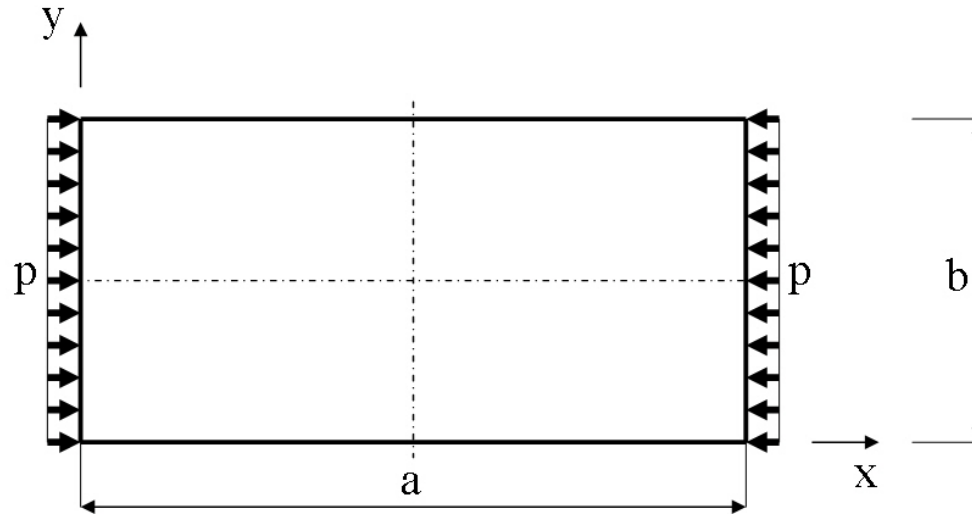


Course in ANSYS

Example0505

Example – Plate

**Objective:**

Compute the buckling load

Tasks:

How should this be modelled?

Compare results with results obtained from norm calculations?

Topics:

Element type, Real constants, modeling, plot results, output graphics

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

$$\nu = 0.3$$

$$a = 200 \text{ mm}$$

$$b = 100 \text{ mm}$$

$$t = 1 \text{ mm}$$

$$p = ?$$

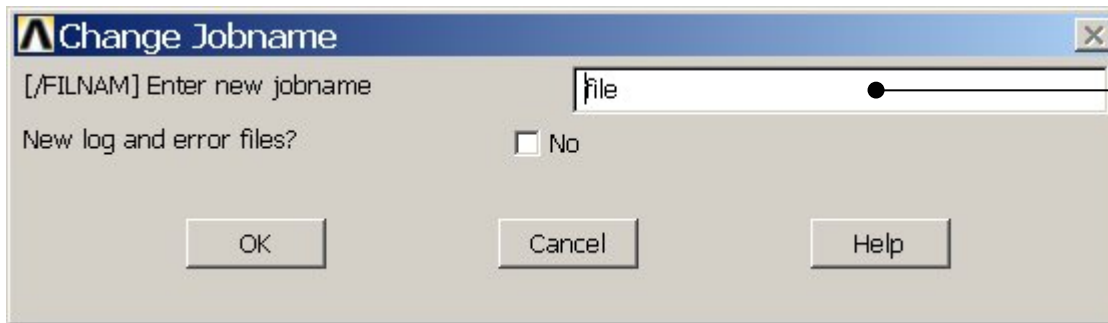
Example - title

Utility Menu > File > Change Jobname

/jobname, Example0505

GUI

Command line entry

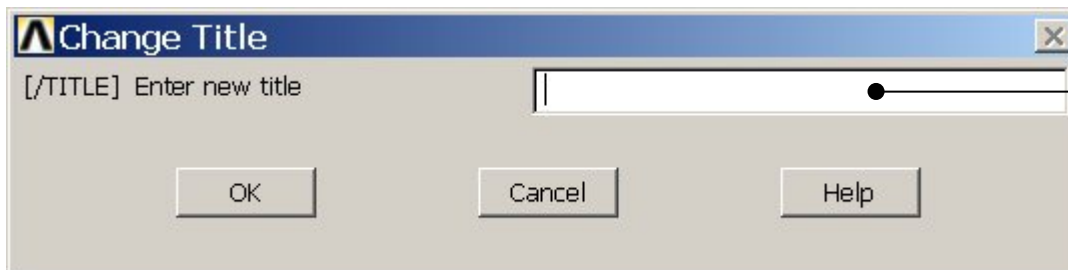


Enter: Example0505

Utility Menu > File > Change Title

/title, Plate

Enter: Plate



Example – Areas Rectangle

Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

Create an area given by $X=(0,200)$ and $Y=(0,100)$

The image shows the ANSYS Main Menu on the left and the 'Create Rectangle by Dimensions' dialog box in the center. The dialog box has a title bar with the ANSYS logo and the text 'Create Rectangle by Dimensions'. Below the title bar, it says '[RECTNG] Create Rectangle by Dimensions'. There are two rows of input fields: 'X1,X2 X-coordinates' and 'Y1,Y2 Y-coordinates'. Each row has two input boxes. Arrows point from text labels to these input boxes: 'Enter 0 or leave empty' points to the first box of the X-coordinates row, 'Enter 200' points to the second box of the X-coordinates row, 'Enter 0 or leave empty' points to the first box of the Y-coordinates row, and 'Enter 100' points to the second box of the Y-coordinates row. At the bottom of the dialog box are four buttons: 'OK', 'Apply', 'Cancel', and 'Help'. An arrow points from the text 'Press OK' to the 'OK' button. The ANSYS Main Menu on the left shows a tree structure with 'Preprocessor' expanded, and 'Modeling' > 'Create' > 'Areas' > 'Rectangle' > 'By Dimensions' selected. Below the dialog box, there is a note: 'Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)'. At the bottom of the slide, the text 'Example0505' and 'Computational mechanics, AAU, Esbjerg' are visible, along with the page number '4'.

Enter 0 or leave empty

Enter 200

Enter 0 or leave empty

Enter 100

Press OK

Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

Example0505

Computational mechanics, AAU, Esbjerg

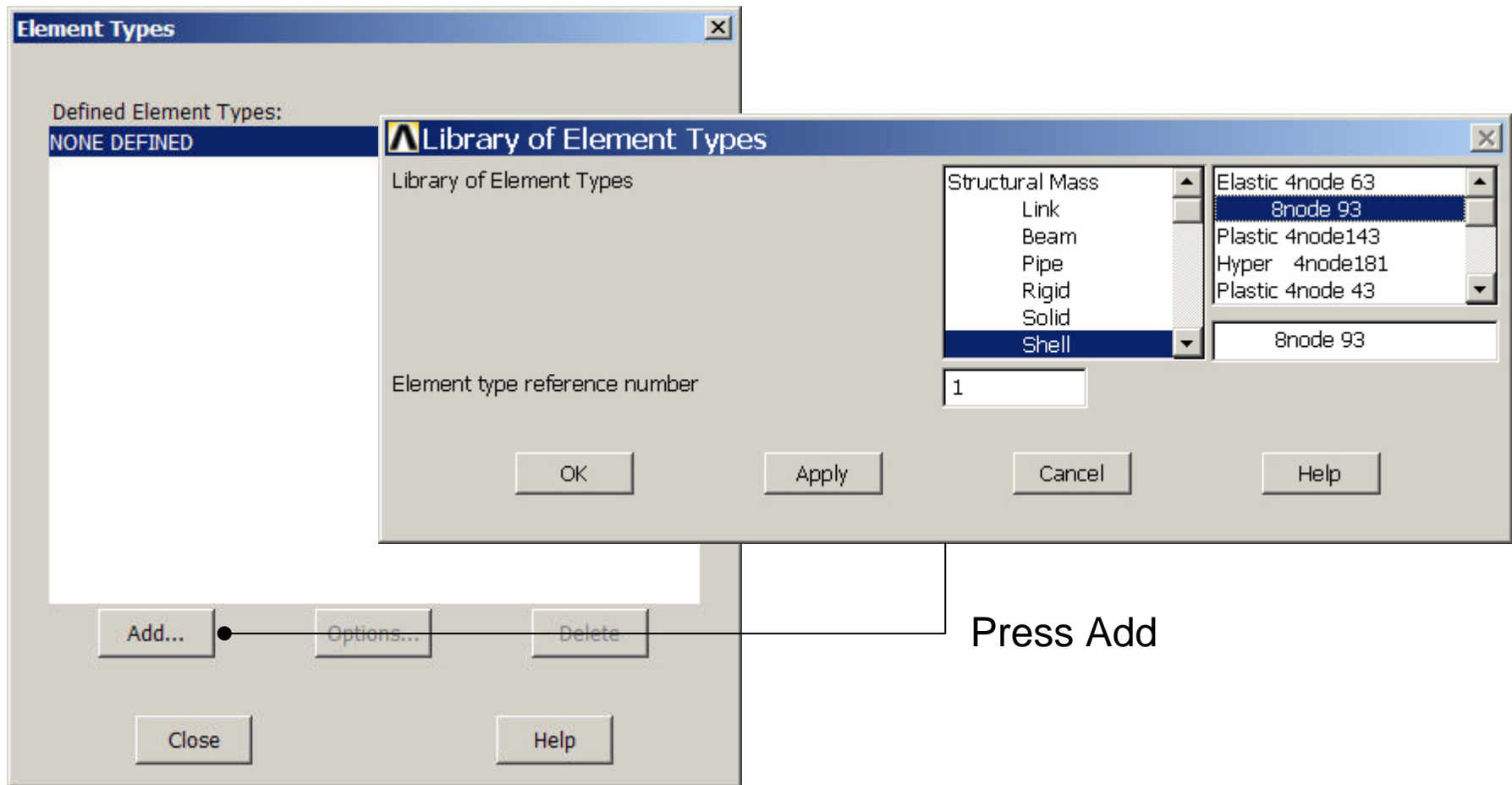
4

Example – Areas Rectangle



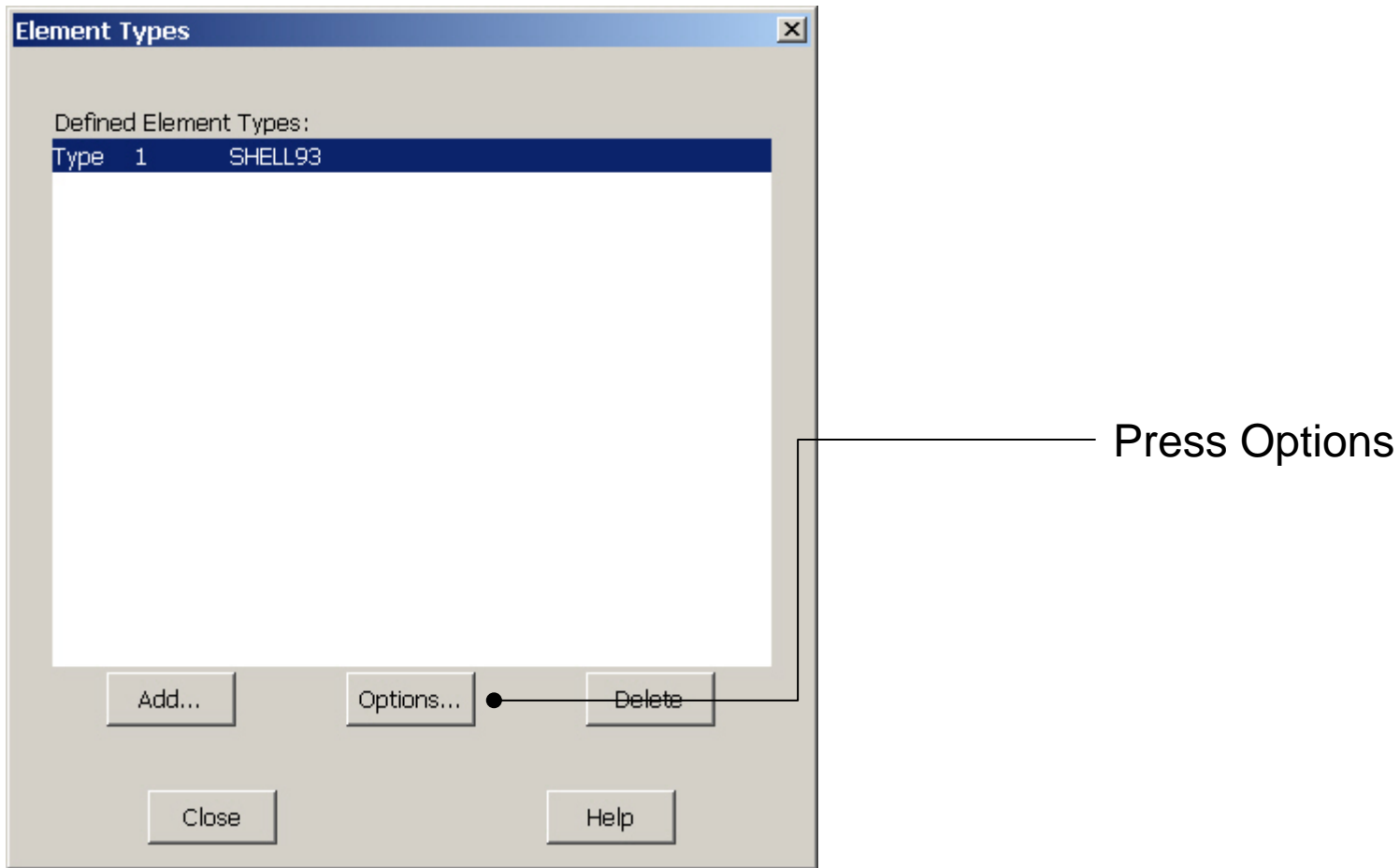
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



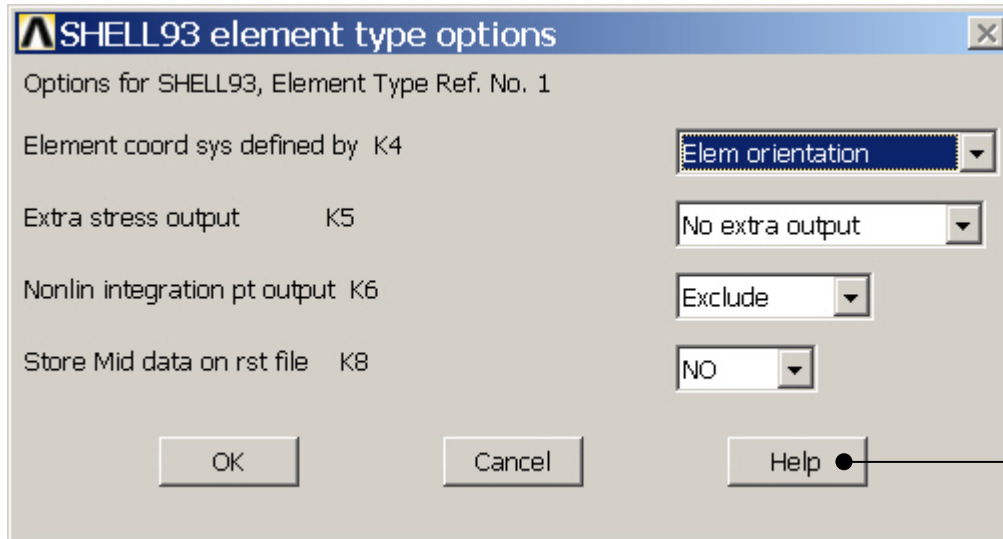
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

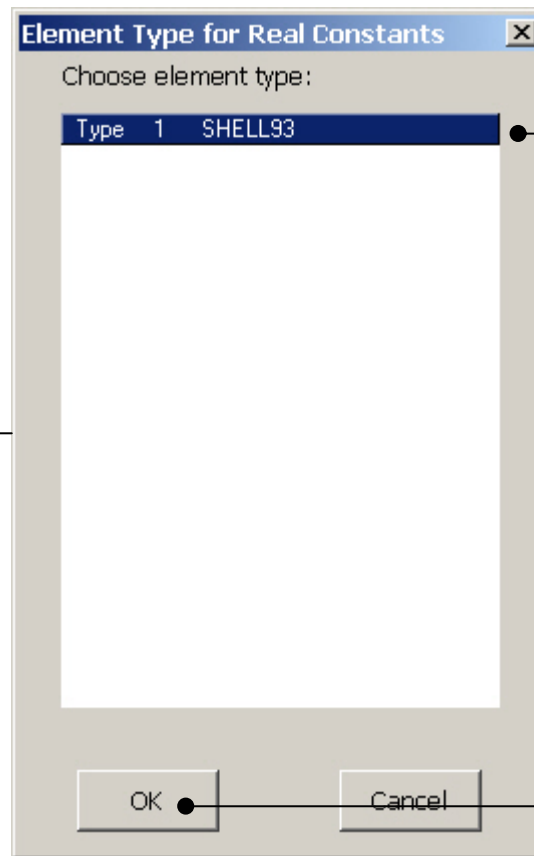
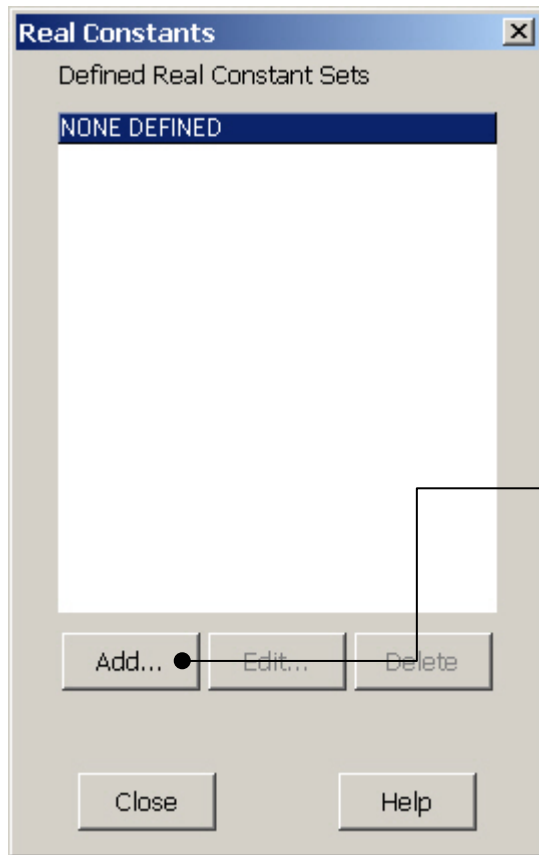
Preprocessor > Element Type > Add/Edit/Delete



Press Help to learn more about the element.

Example – Real Constants

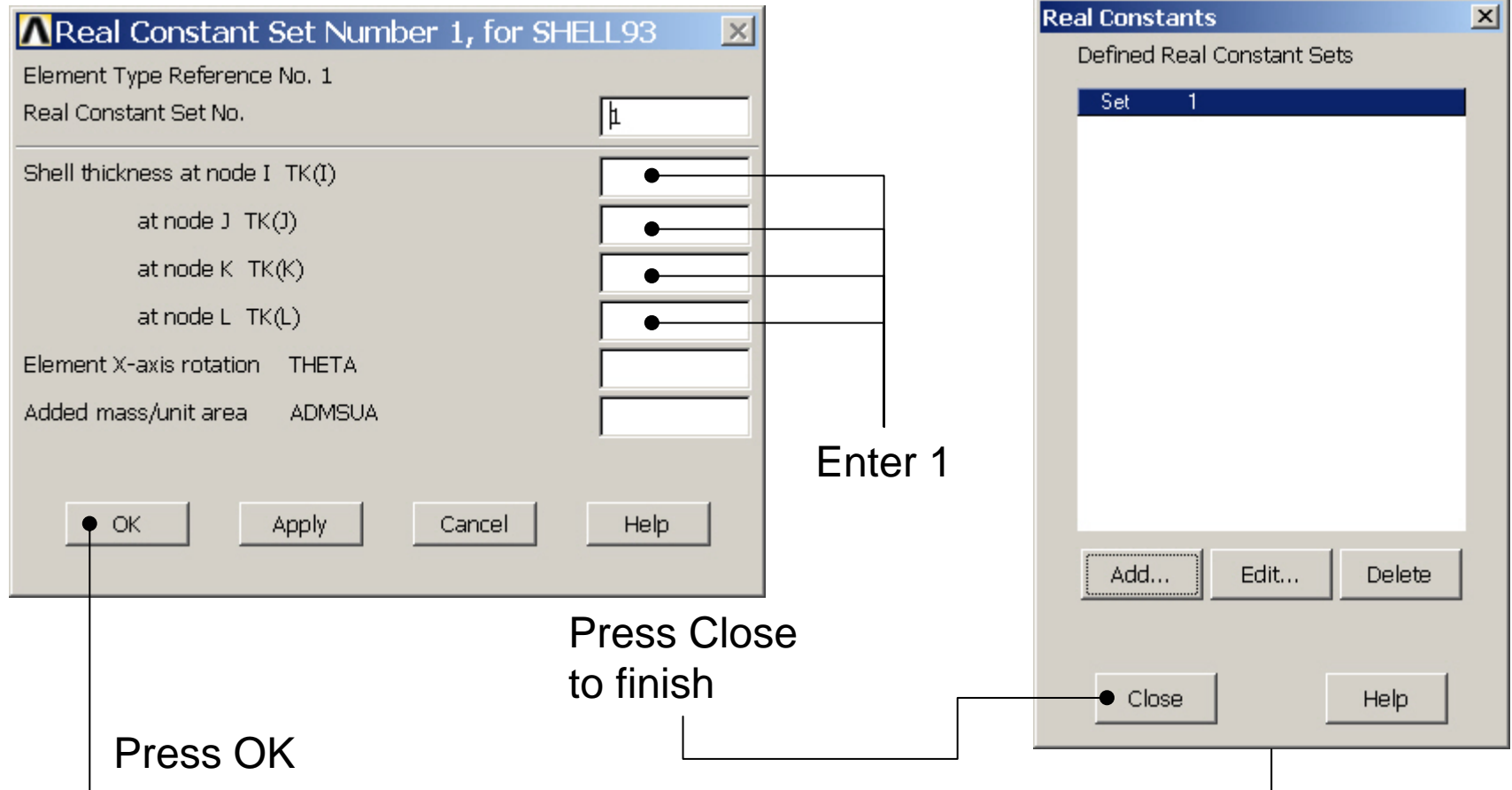
Preprocessor > Real Constants > Add



Place the cursor on the relevant element and press OK

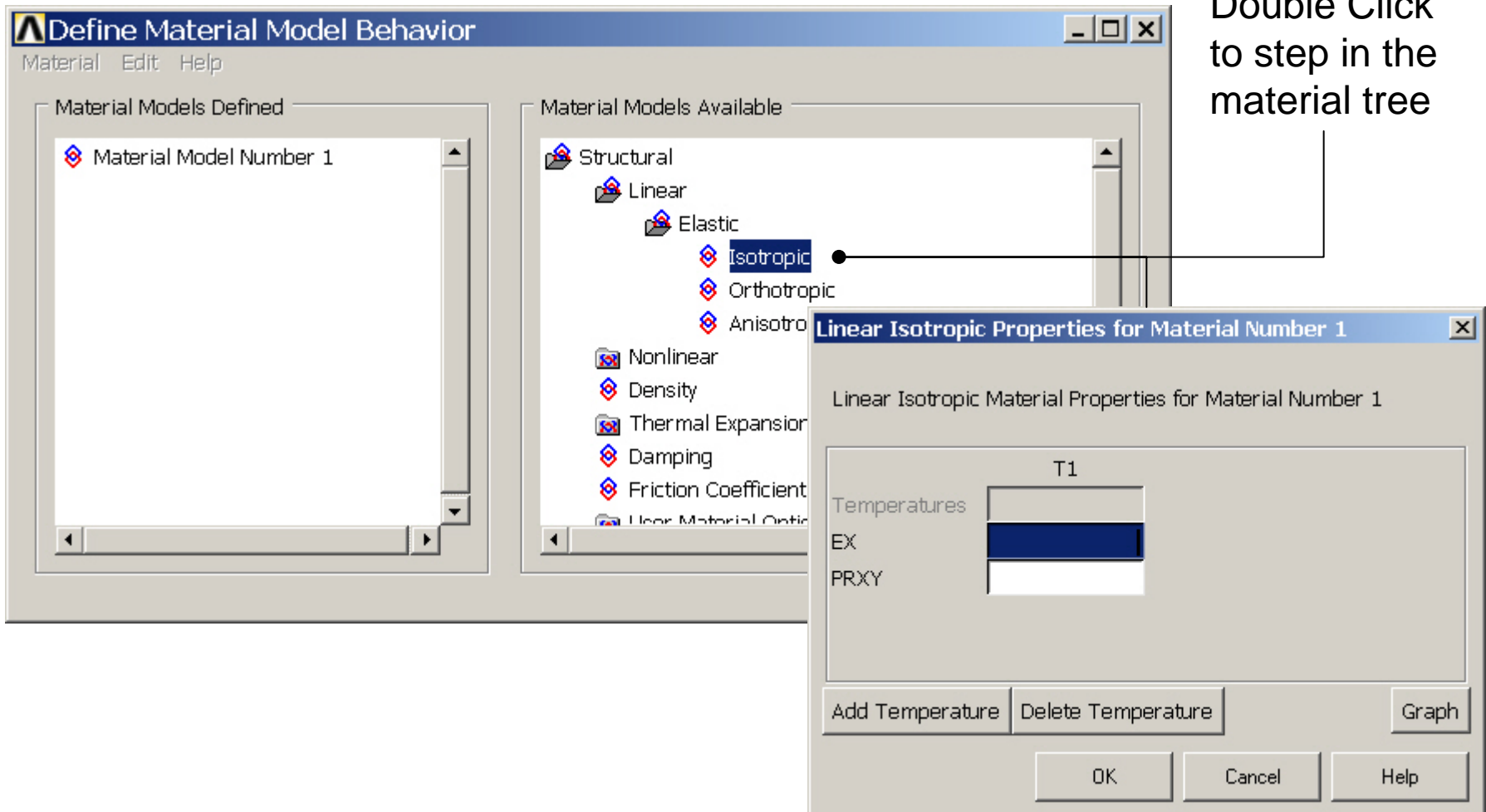
Example - Real Constants

Preprocessor > Real Constants > Add



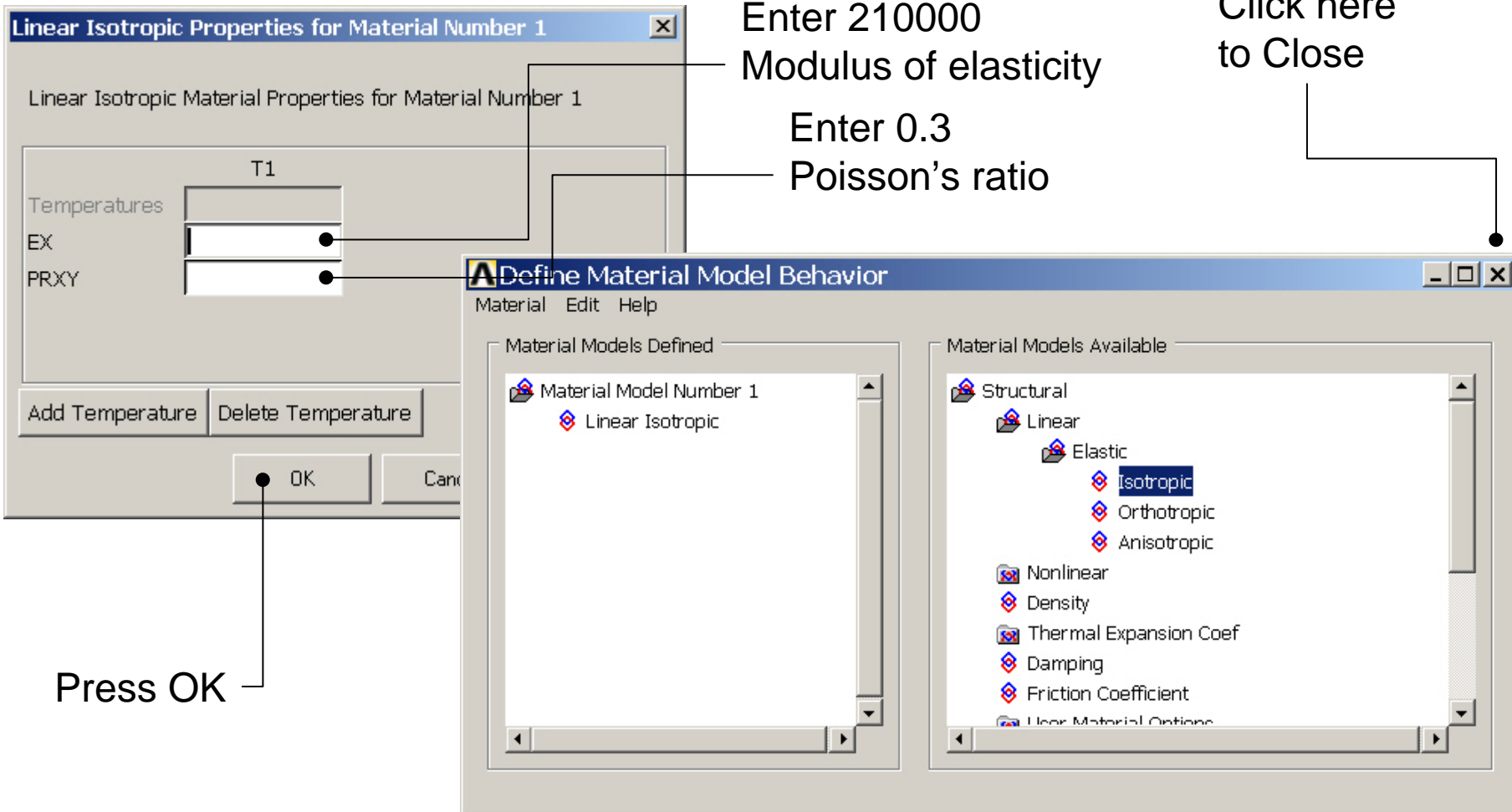
Example - Material Properties

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
the two
shortest 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 10

Example - Meshing

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

Select/Pick
the two
longest 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

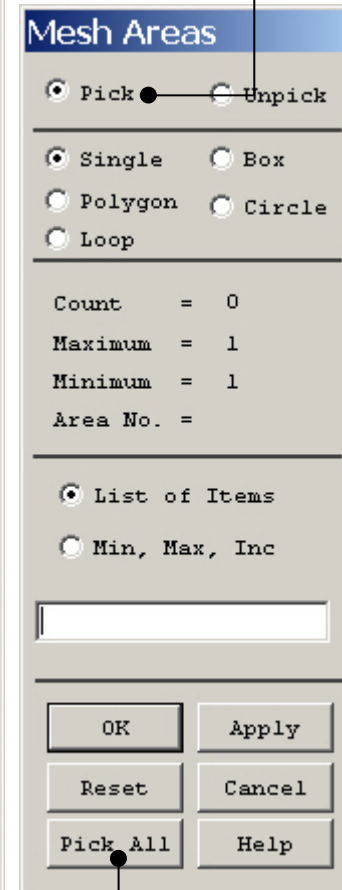
OK Apply Cancel Help

Press OK when finish with selection

Enter 20

Example - Meshing

Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

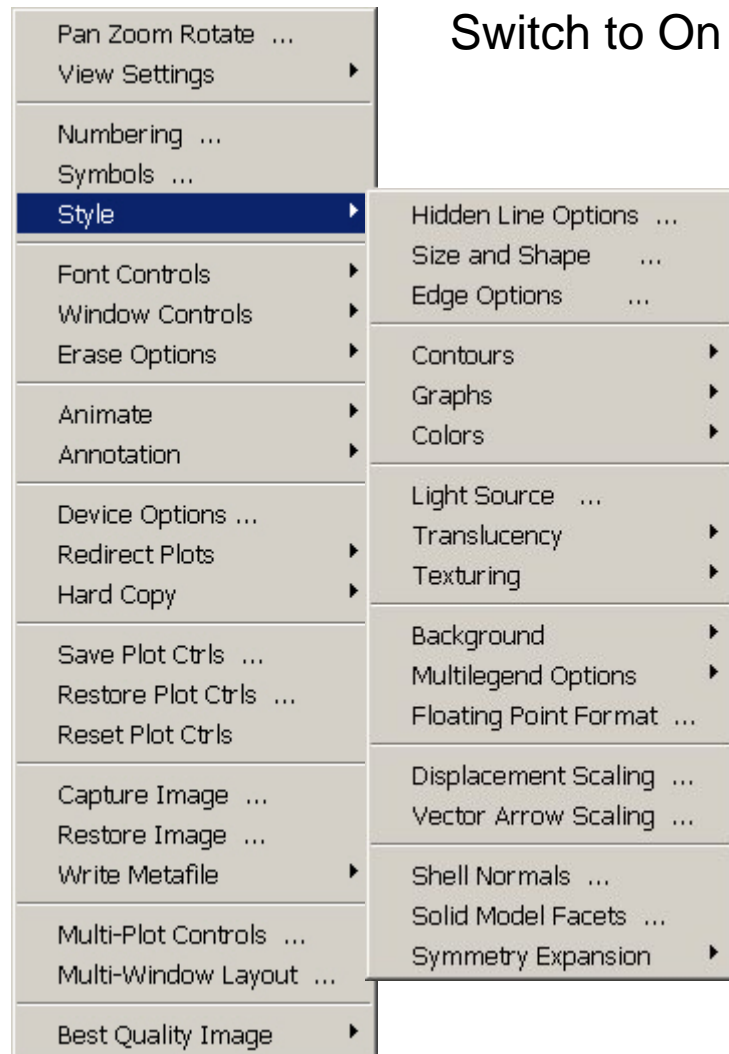


Select individual areas to be meshed

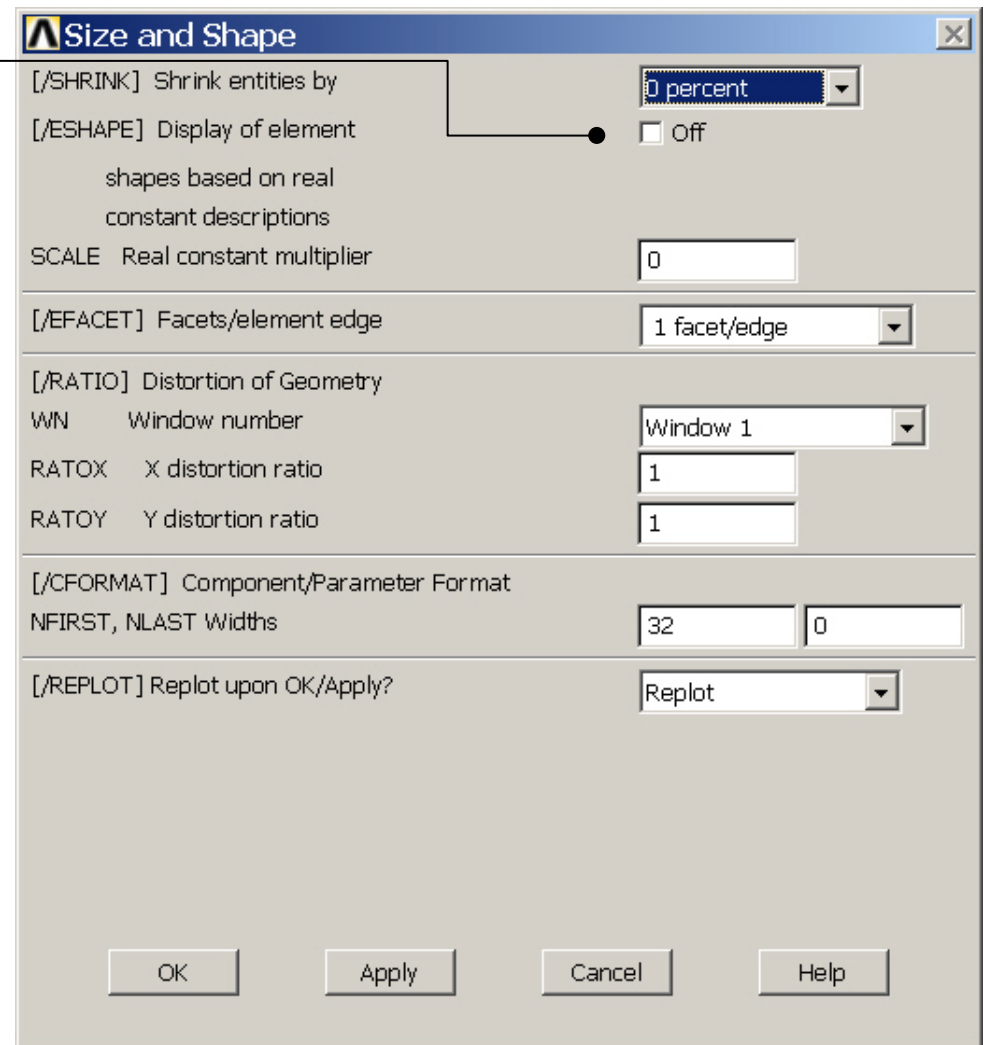
NB: It is often necessary to “Clear” the model for example if Element Type or model geometry is to be changed

Select all areas defined to be meshed

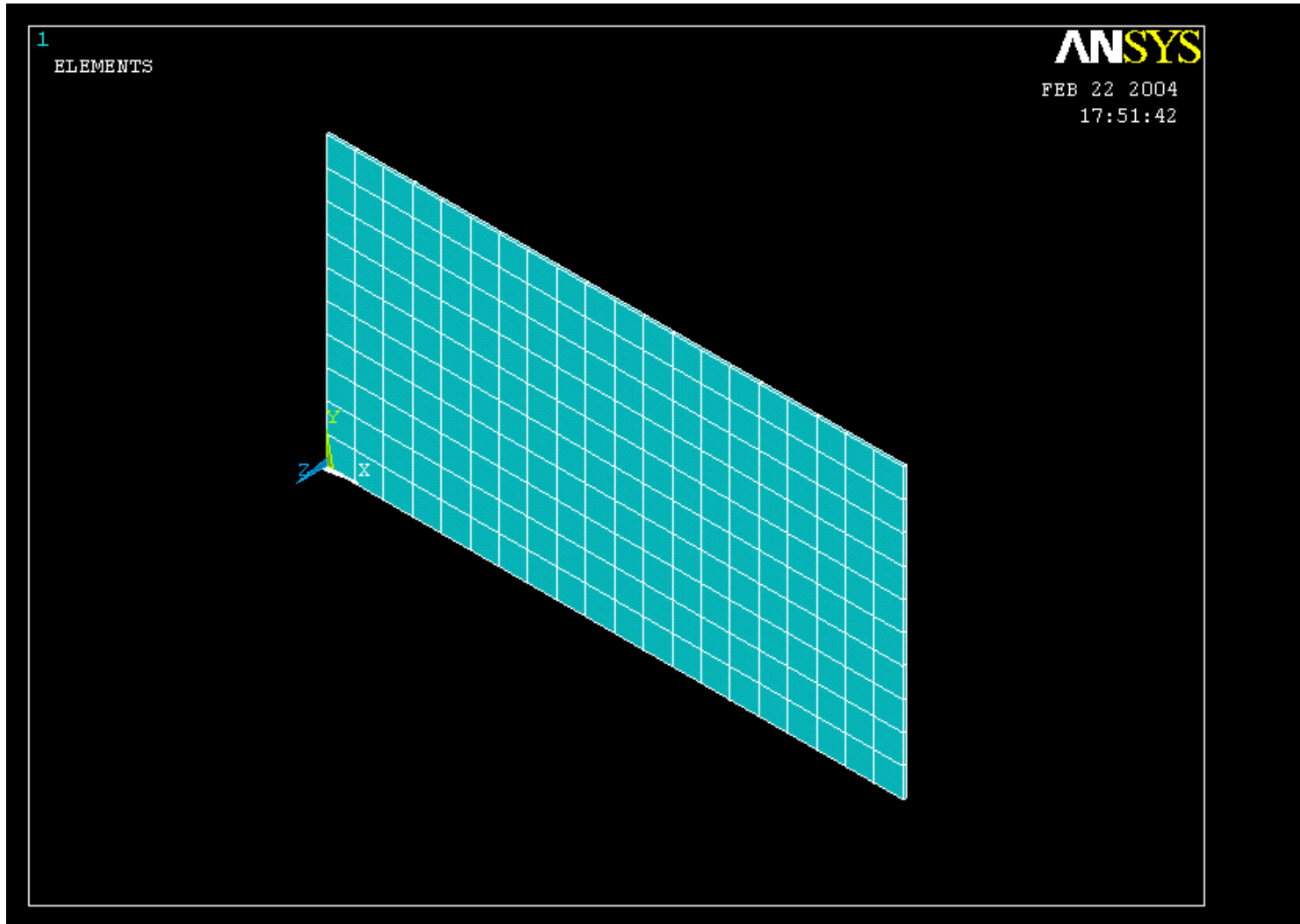
Example - PlotCtrls Menu



Switch to On

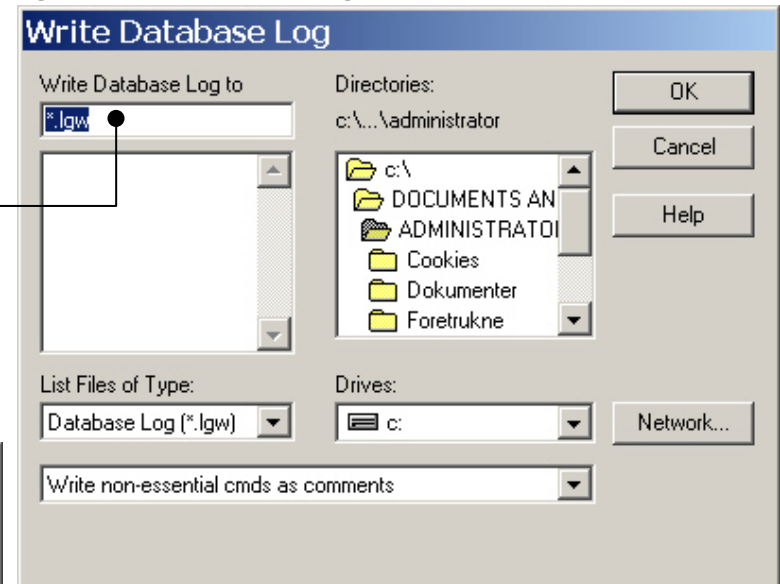


Example – Display of Element

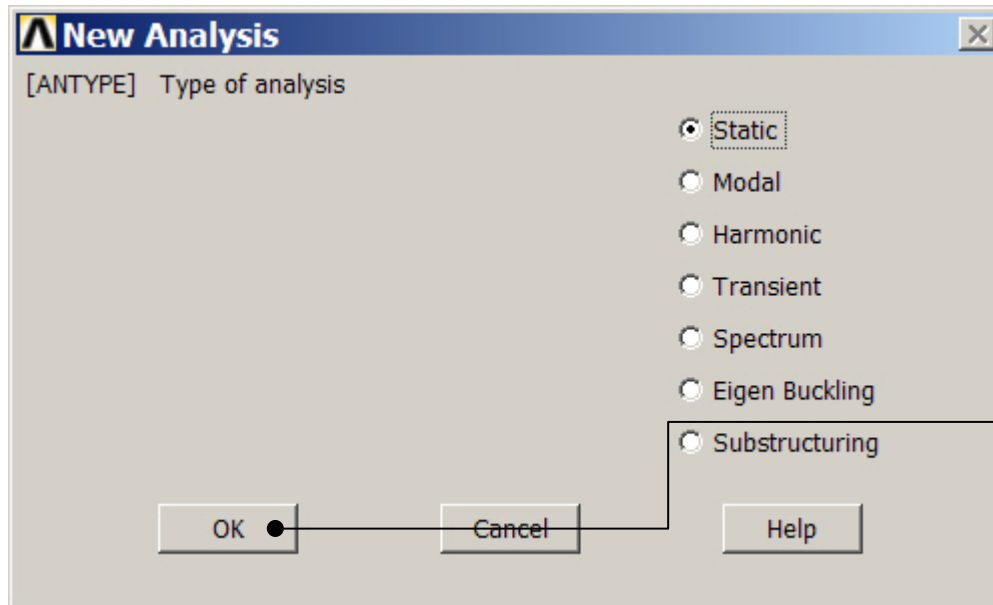


Example – Analysis Type

File > Write DB log file
Enter “example0505.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 - [0]
- 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

Buttons: OK, Cancel, Help

Annotations:

- A line points from the 'Unabridged Menu' option in the ANSYS Main Menu to the 'Static or Steady-State Analysis' dialog box.
- A line points from the 'Analysis Options' option in the ANSYS Main Menu to the 'Static or Steady-State Analysis' dialog box.
- A line points from the 'Stress stiffness or prestress' dropdown menu to the text 'Select Prestress ON'.

Activate the
Unabridged menu

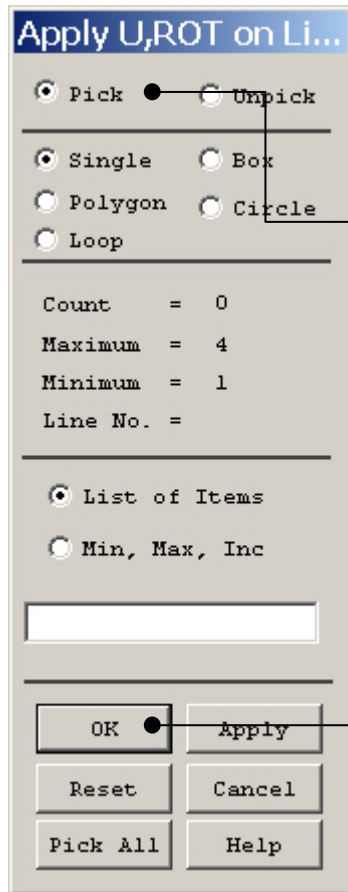
Select Prestress ON

Example0505

19

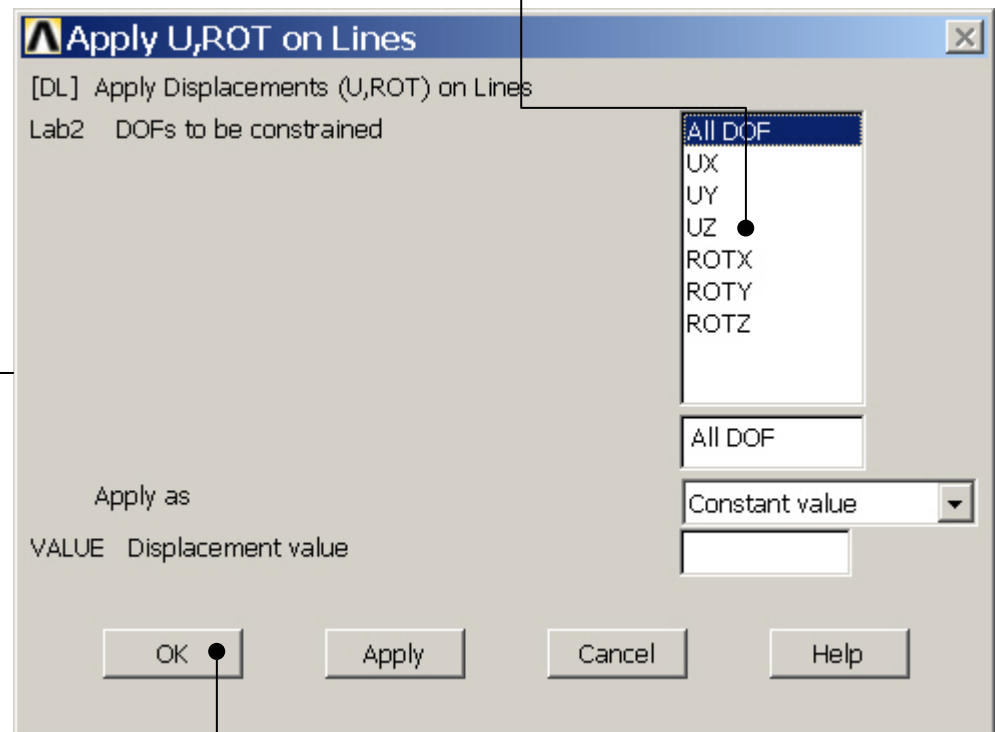
Example – Define Loads

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



Select the
bottom straight
line

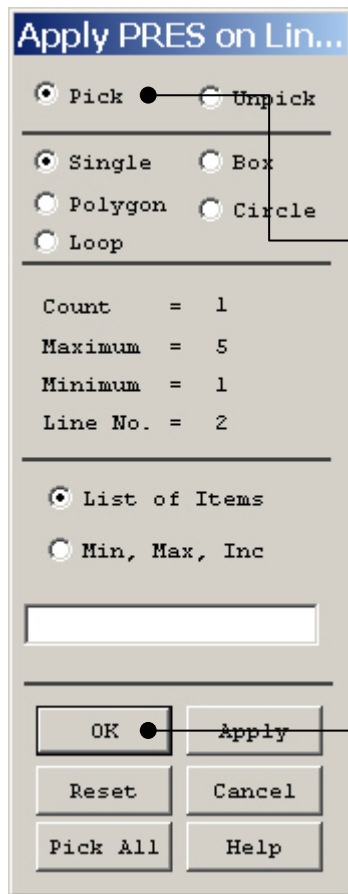
Select UZ to fix the plate in the z-direction



Press OK

Example – Define Loads

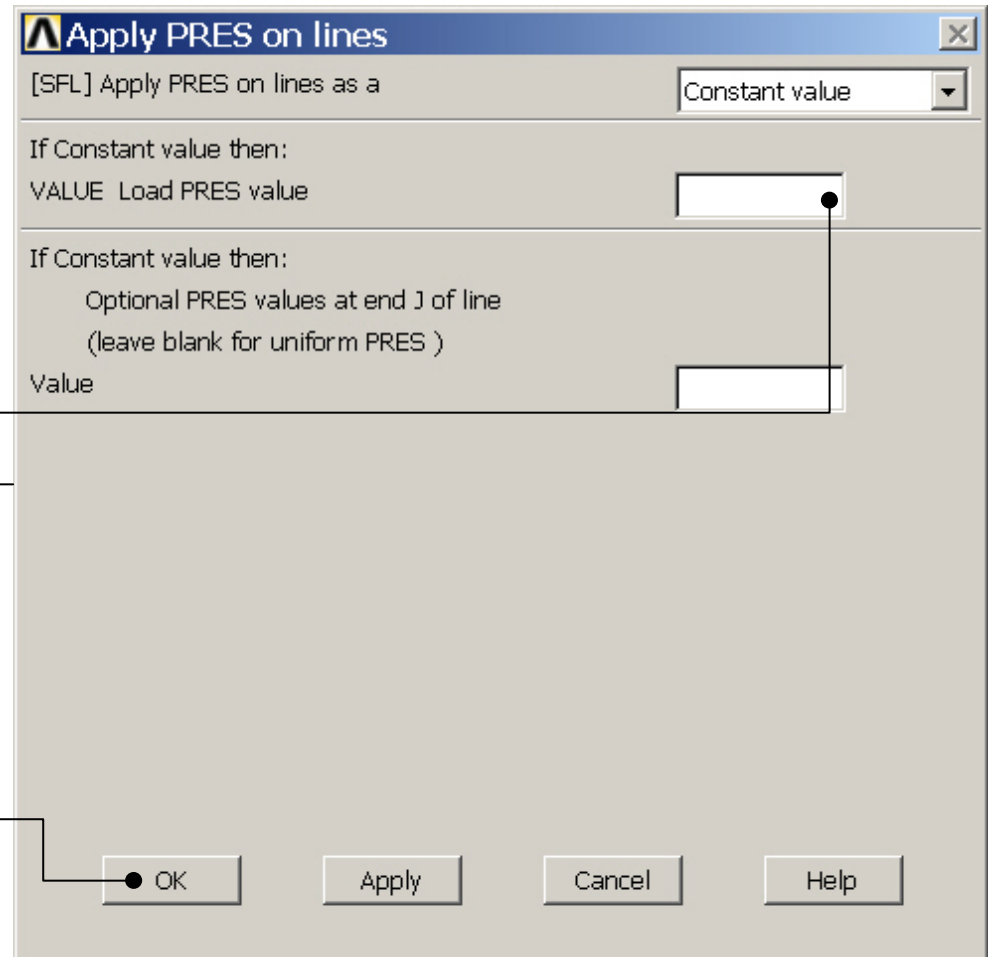
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the
right
straight line

Enter -1

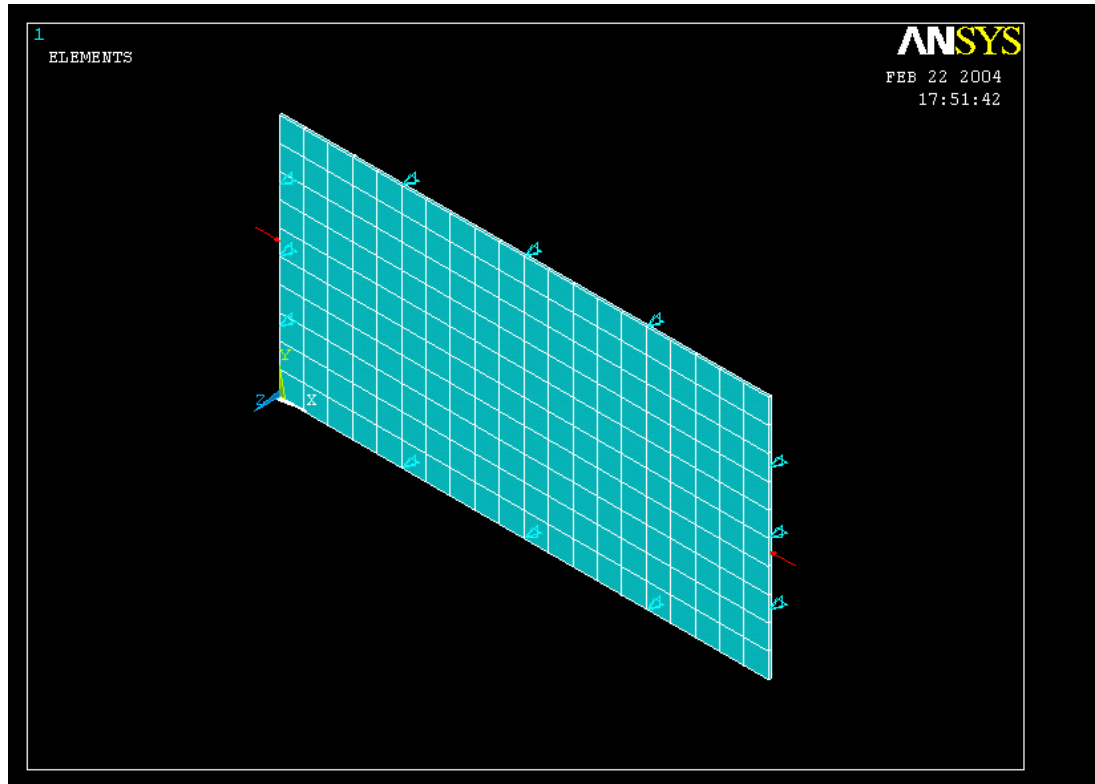
Press OK
to finish



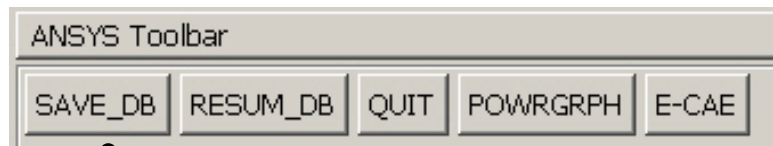
Example0505

Note: Pressure acts normal and
inward to a surface
ANSYS
Computational Mechanics, AAU, Esbjerg

Example - Save



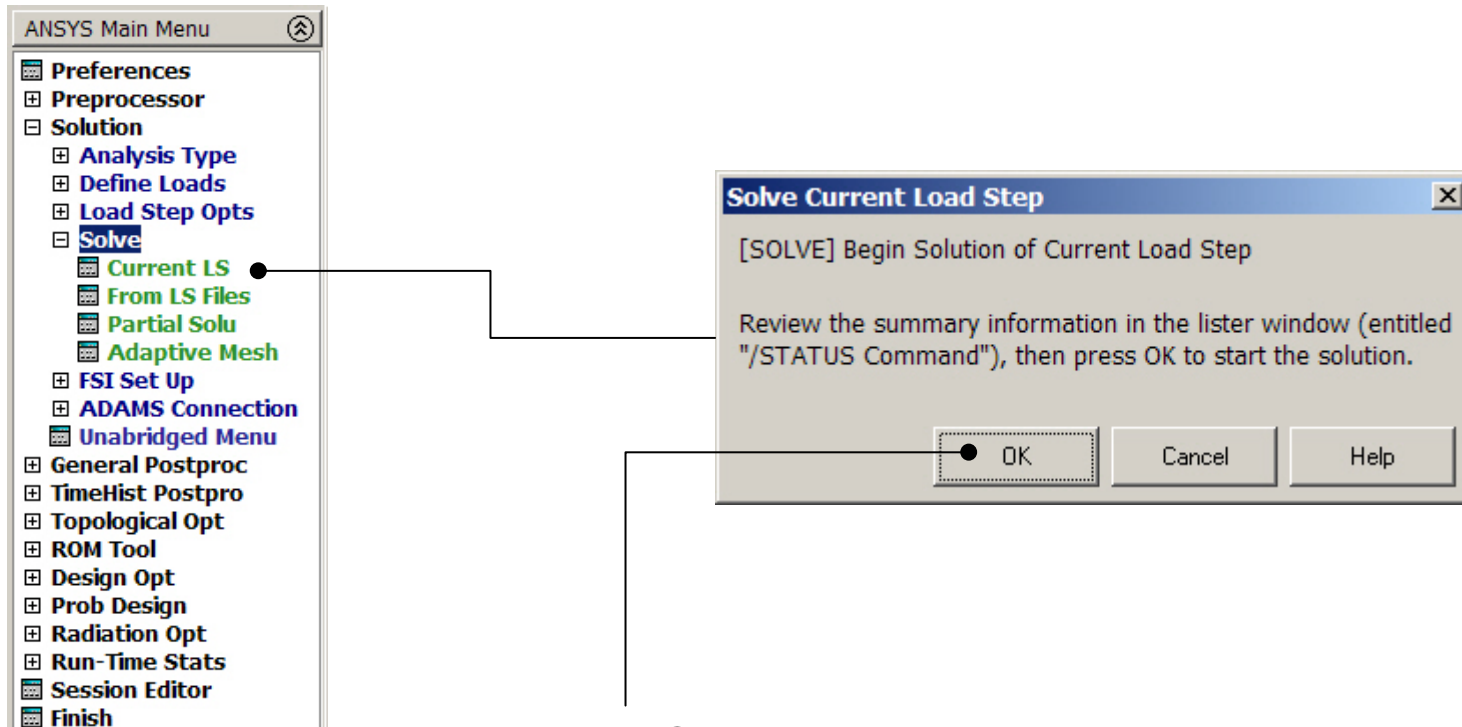
Display of Analysis model



Save the model

Example - Solve

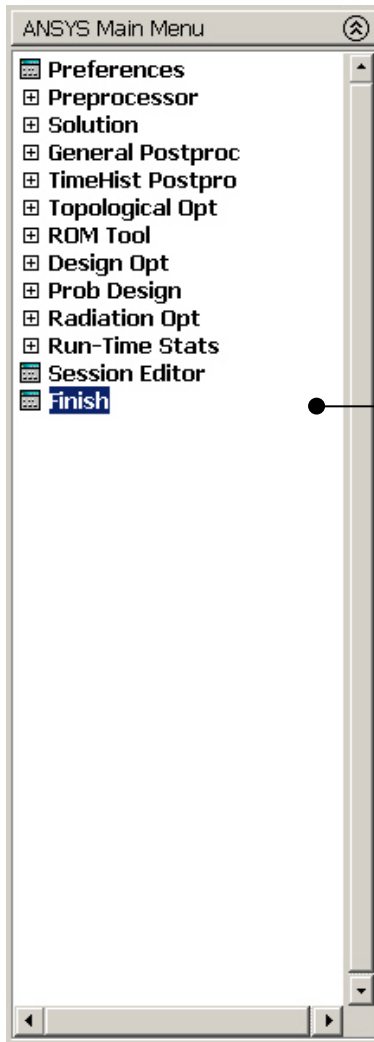
Solution > Solve > Current LS



Press OK

Example0505

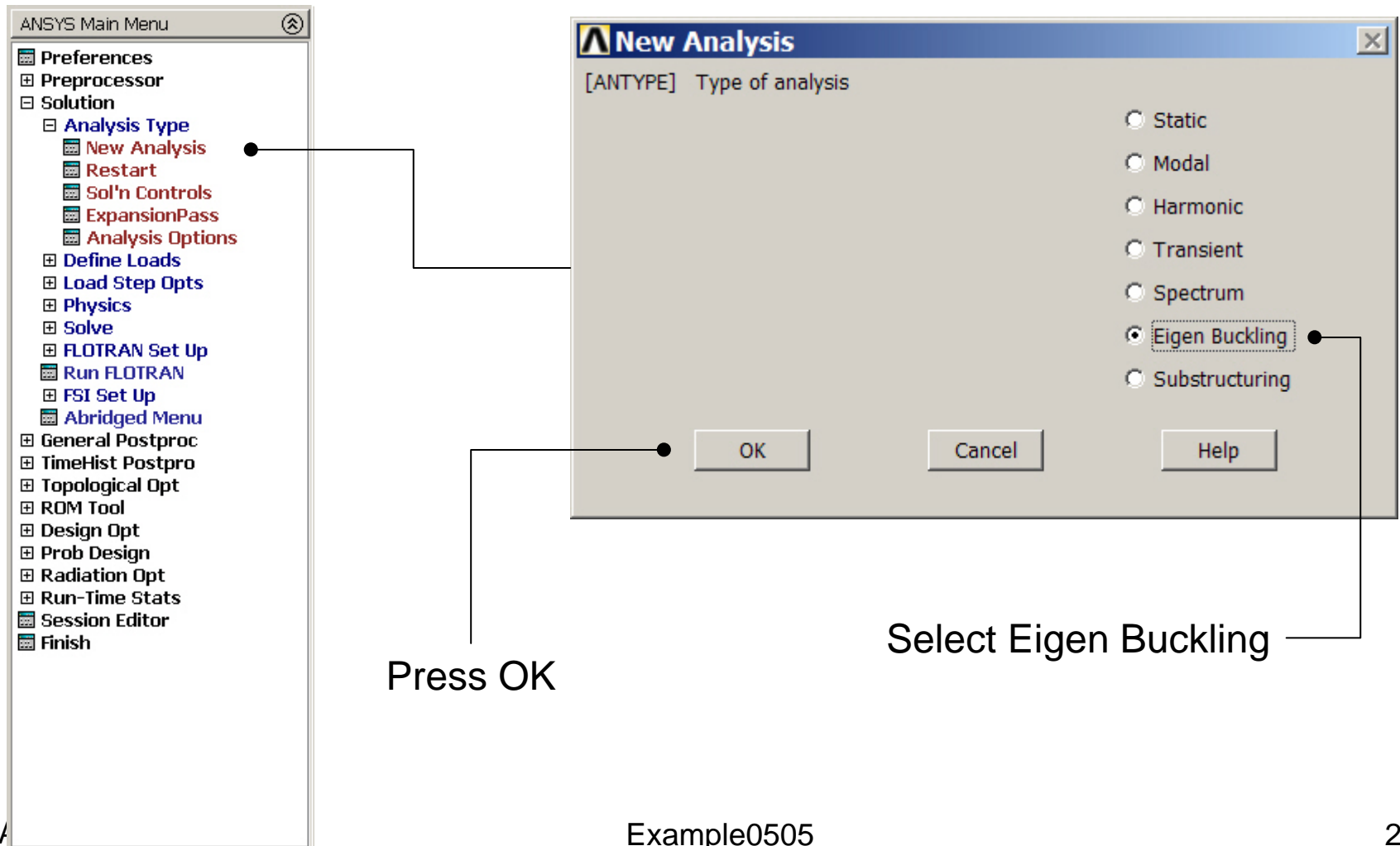
Example - Finish



Press Finish to end the static solution

Eigen Buckling - New Analysis

Main Menu> Solution> Analysis Type> New Analysis

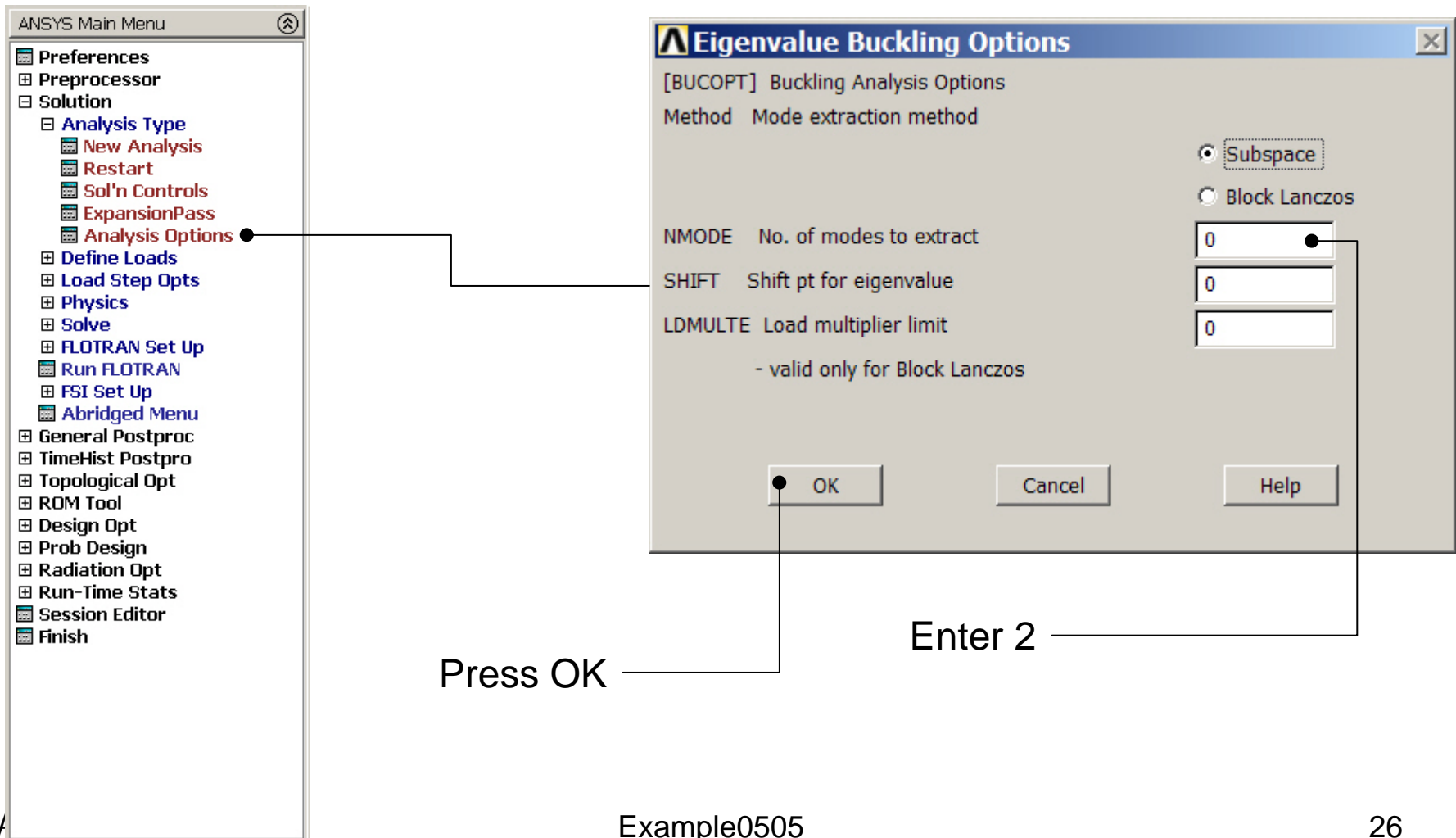


Example0505

25

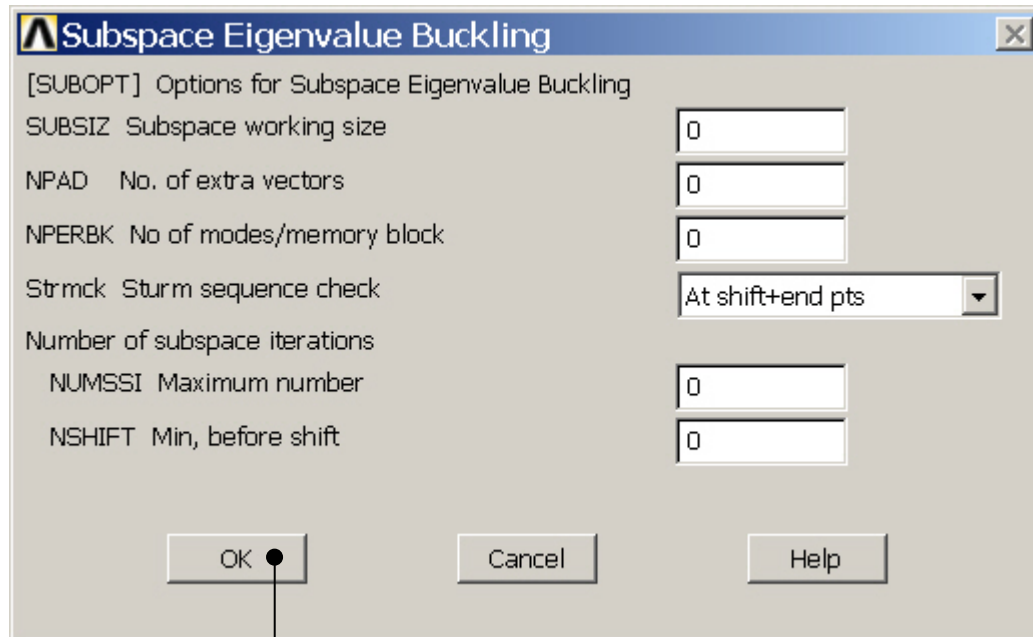
Eigen Buckling – Analysis Options

Main Menu> Solution> Analysis Type> Analysis Options



Example0505

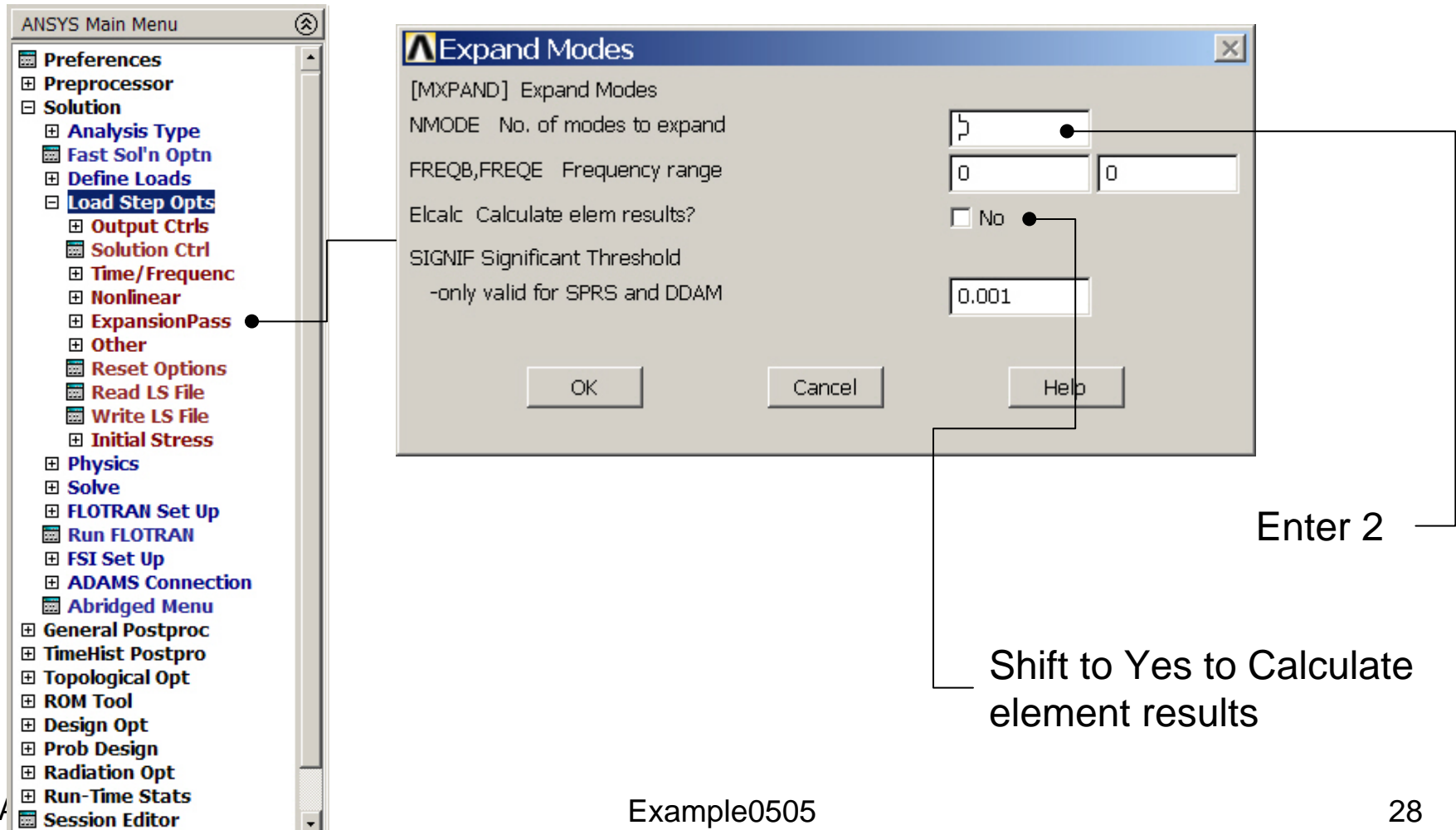
Example – Subspace Options



Press OK

Eigen Buckling – Expanding Modes

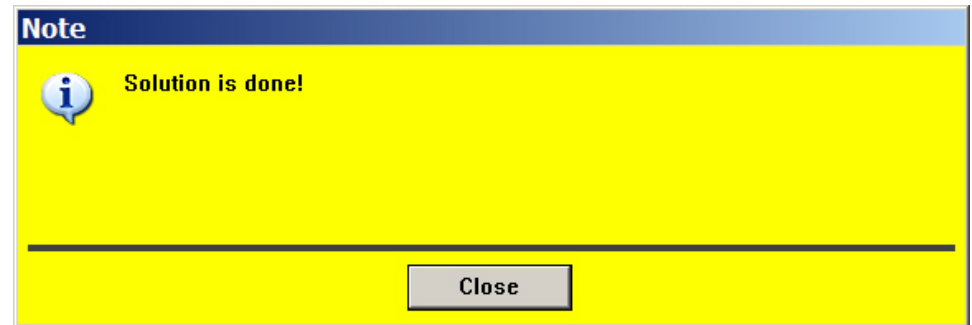
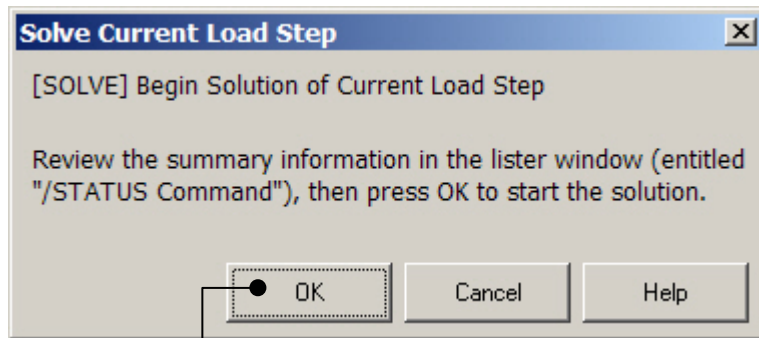
Main Menu> Solution> Load Step Opts > ExpansionPass >
Single Expand > Expand Modes



Example0505

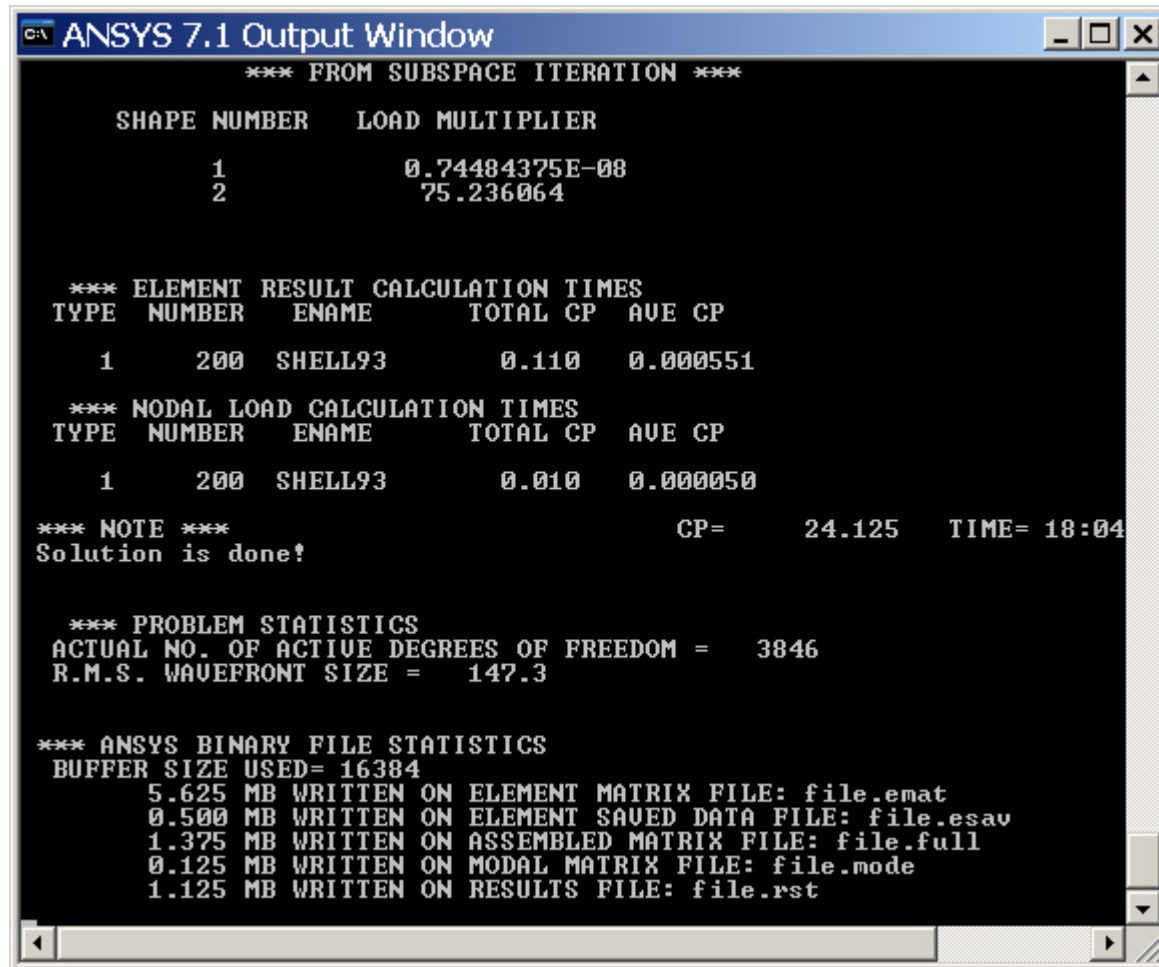
Example - Solve

Solution > Solve > Current LS



Press OK

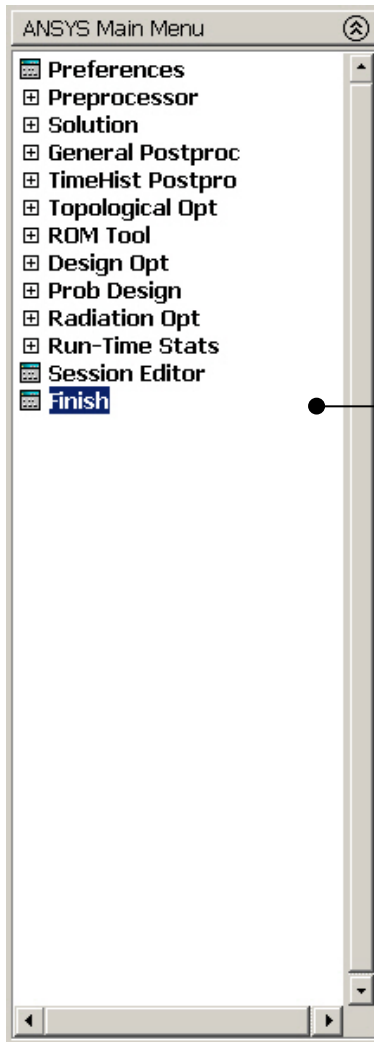
Example – Output Window



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:

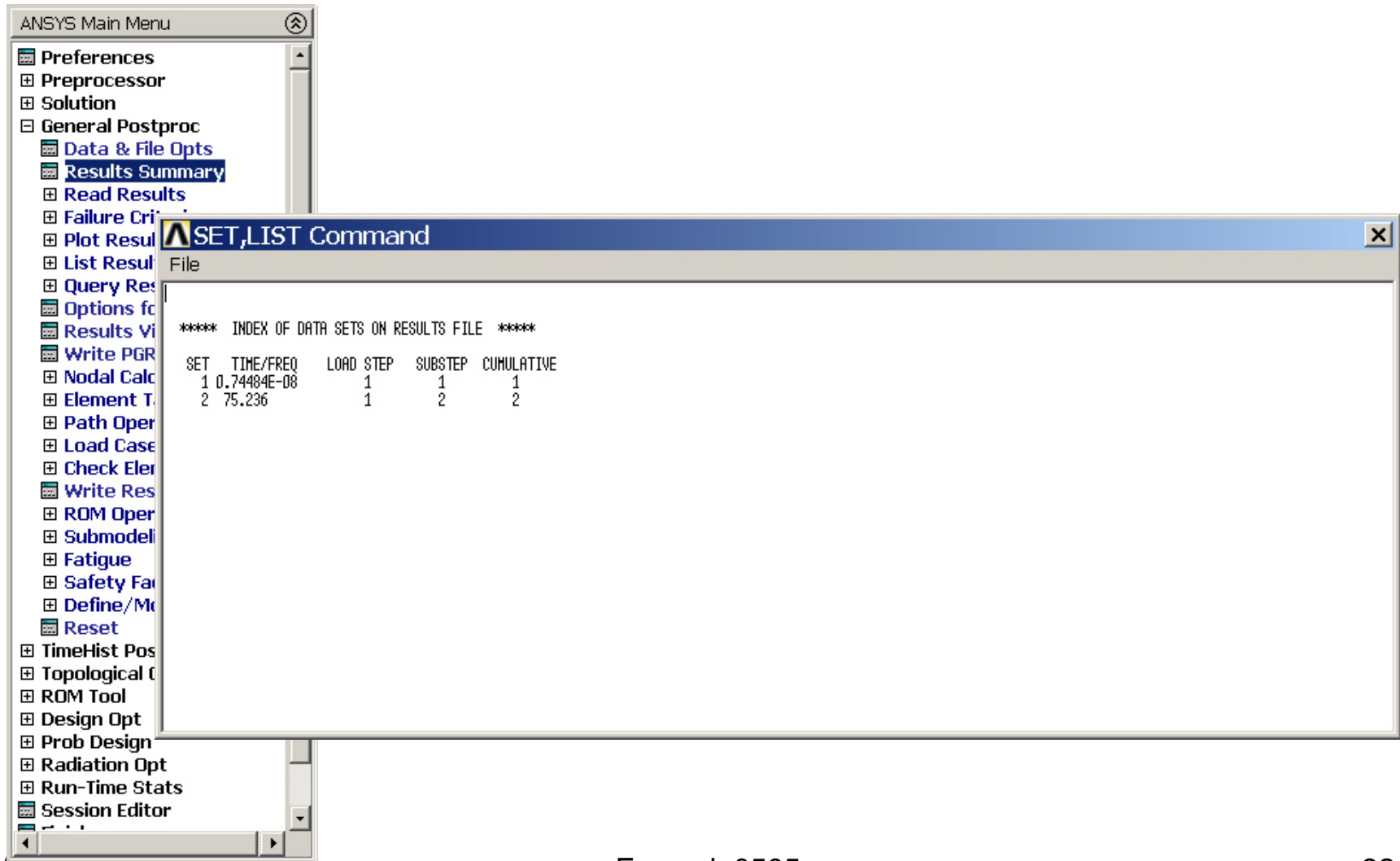
```
*** FROM SUBSPACE ITERATION ***  
  
SHAPE NUMBER    LOAD MULTIPLIER  
      1          0.74484375E-08  
      2          75.236064  
  
*** ELEMENT RESULT CALCULATION TIMES  
TYPE  NUMBER  ENAME      TOTAL CP  AVE CP  
   1     200  SHELL93      0.110    0.000551  
  
*** NODAL LOAD CALCULATION TIMES  
TYPE  NUMBER  ENAME      TOTAL CP  AVE CP  
   1     200  SHELL93      0.010    0.000050  
  
*** NOTE ***  
Solution is done!                                CP=      24.125    TIME= 18:04  
  
*** PROBLEM STATISTICS  
ACTUAL NO. OF ACTIVE DEGREES OF FREEDOM =    3846  
R.M.S. WAVEFRONT SIZE =    147.3  
  
*** ANSYS BINARY FILE STATISTICS  
BUFFER SIZE USED= 16384  
5.625 MB WRITTEN ON ELEMENT MATRIX FILE: file.emat  
0.500 MB WRITTEN ON ELEMENT SAVED DATA FILE: file.esav  
1.375 MB WRITTEN ON ASSEMBLED MATRIX FILE: file.full  
0.125 MB WRITTEN ON MODAL MATRIX FILE: file.mode  
1.125 MB WRITTEN ON RESULTS FILE: file.rst
```

Example - Finish



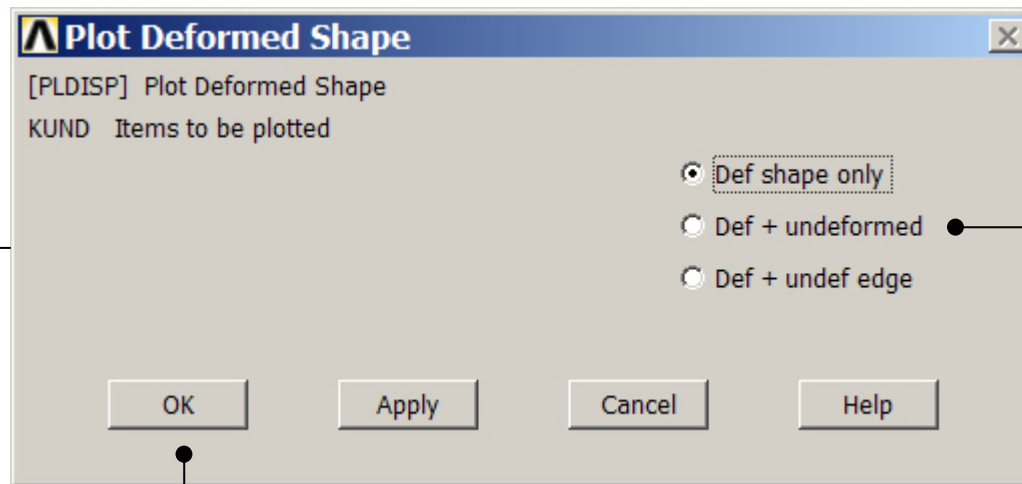
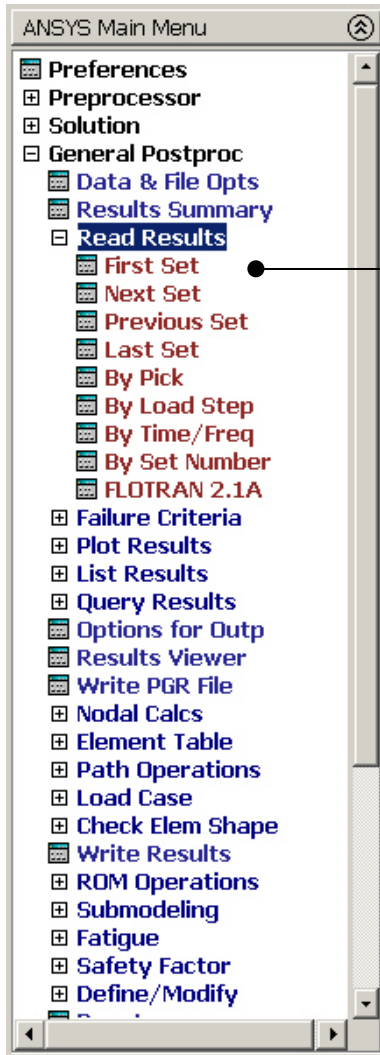
Press Finish to end the eigen buckling solution

Example – Results Summary



Example – Read Results

General Postproc > Plot Results > Deformed Shape

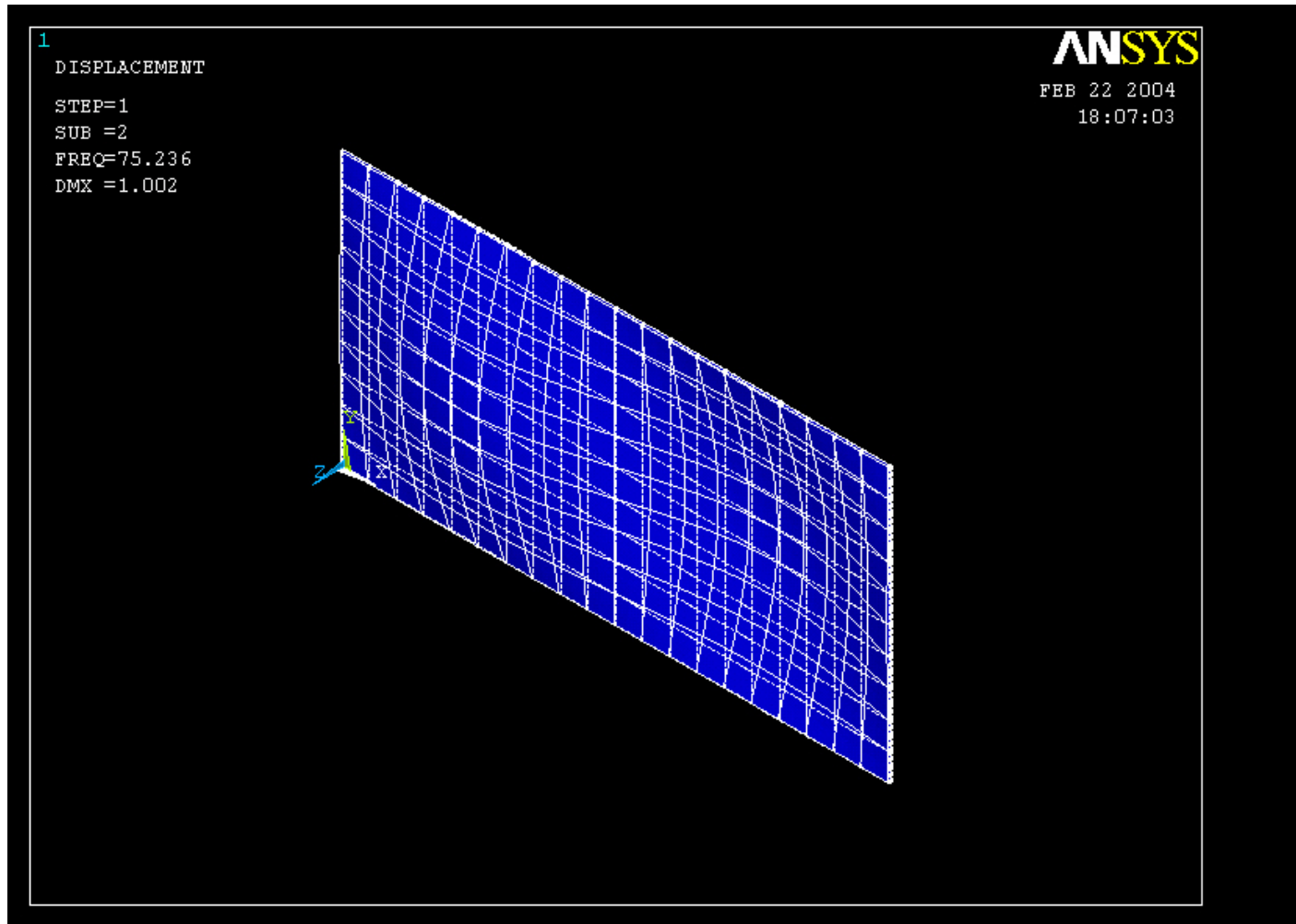


Select "Def+undeformed"
and Press OK

Example – First set

As there are no constraints on rotation about the Z-axis an extra buckling mode of simple rotation at a load of approximately zero

Example – Next set



Example – Contour Plot

