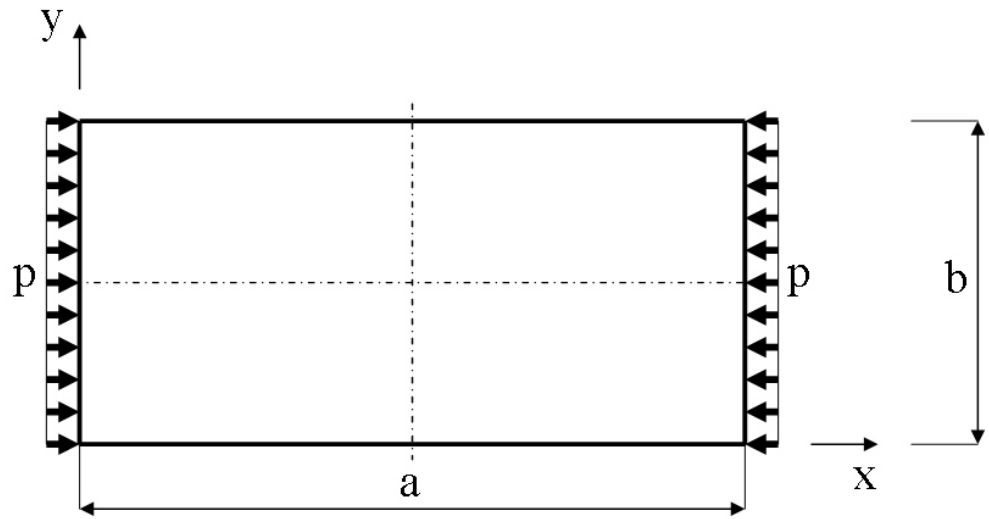
Course in ANSYS

Example0505

Example – Plate



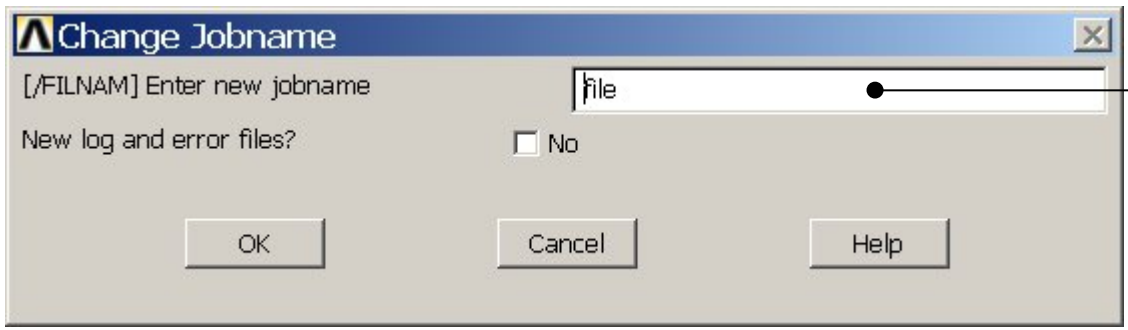
Objective:
Compute the buckling load
Tasks:
How should this be modelled?
Compare results with results obtained from norm calculations?
Topics:
Element type, Real constants, modeling, plot results, output graphics

$E = 210000 N/mm^2$
$\nu = 0.3$
$a = 200mm$
$b = 100mm$
$t = 1mm$
$p = ?$

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0505 — Command line entry

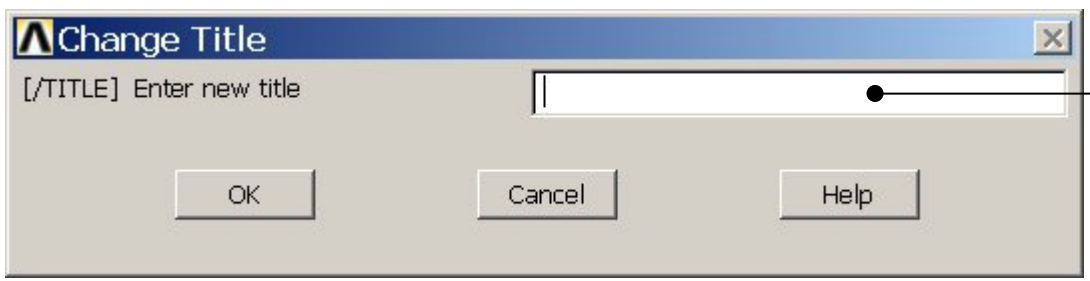


Enter: Example0505

Utility Menu > File > Change Title

/title, Plate

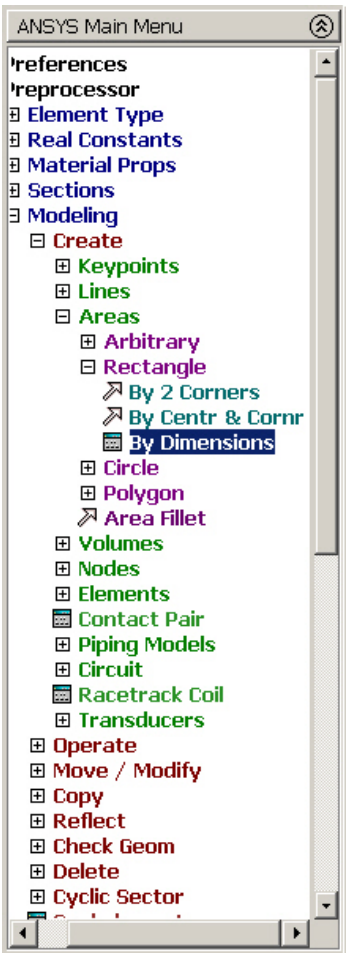
Enter: Plate

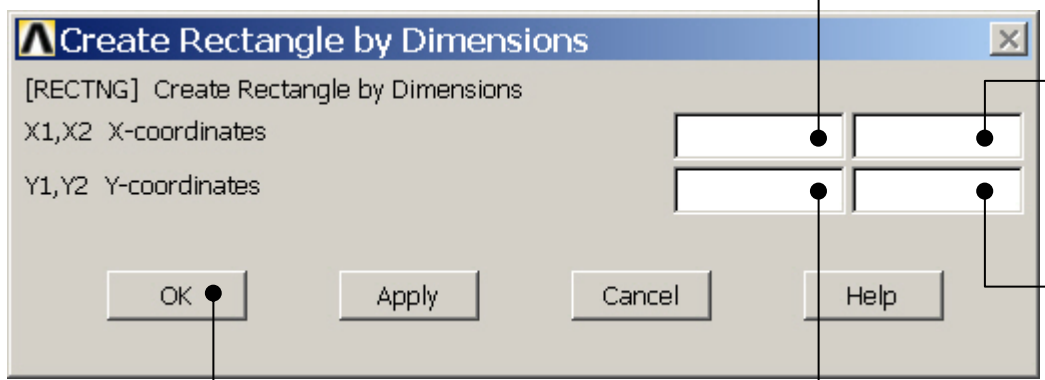


Example – Areas Rectangle

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

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





Enter 0 or leave empty

Enter 200

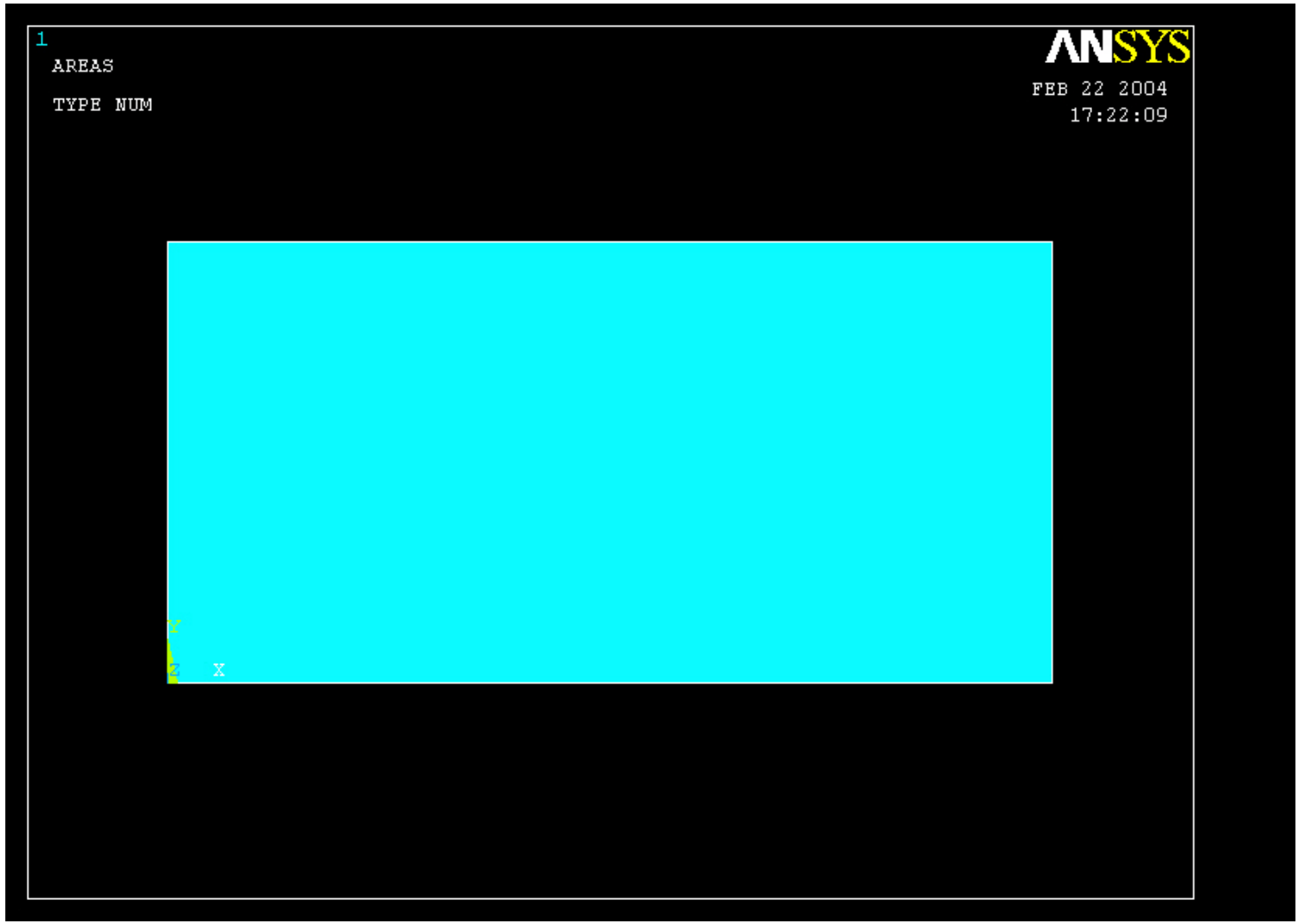
Enter 0 or leave empty

Enter 100

Press OK

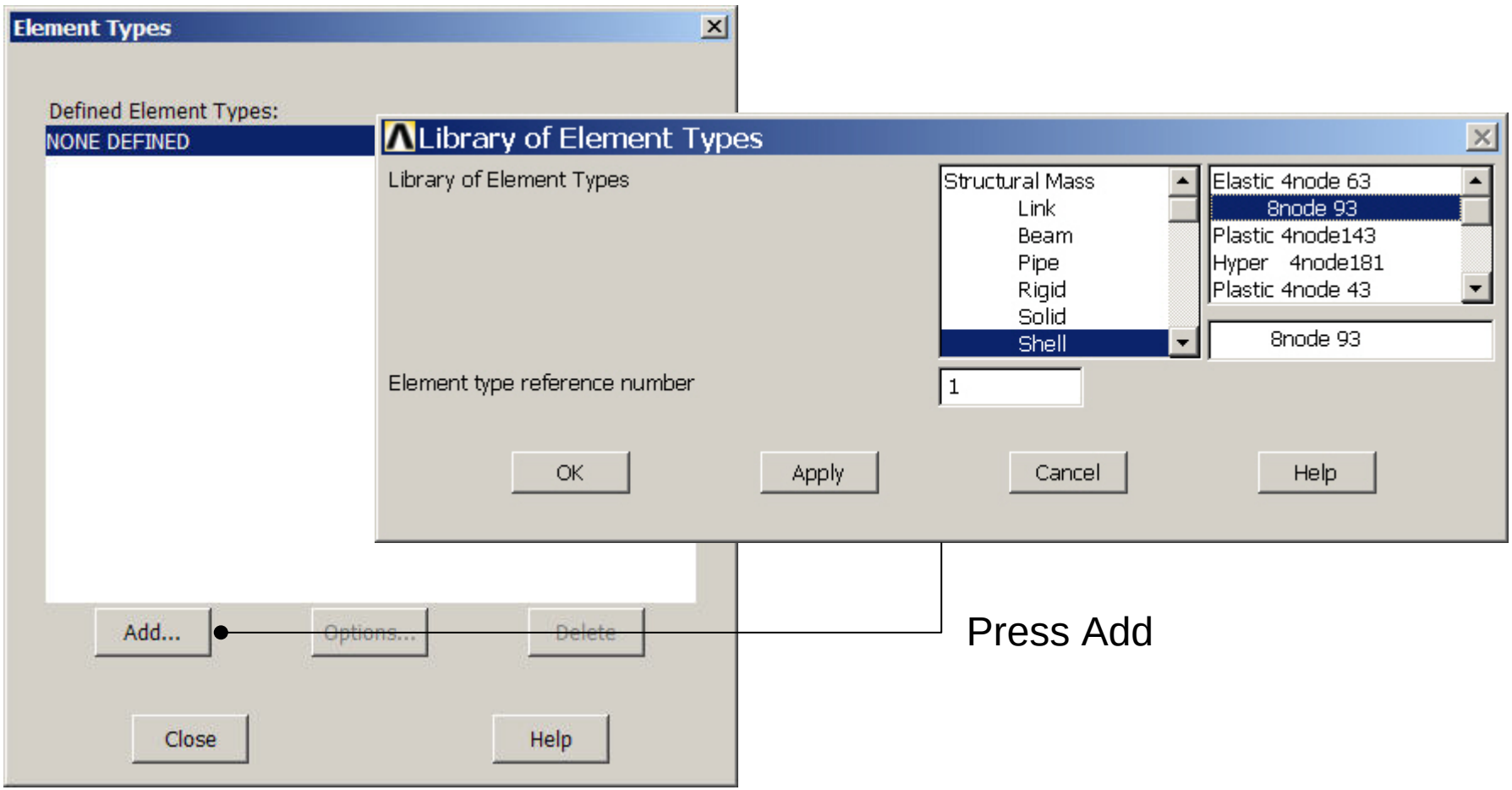
Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

Example – Areas Rectangle



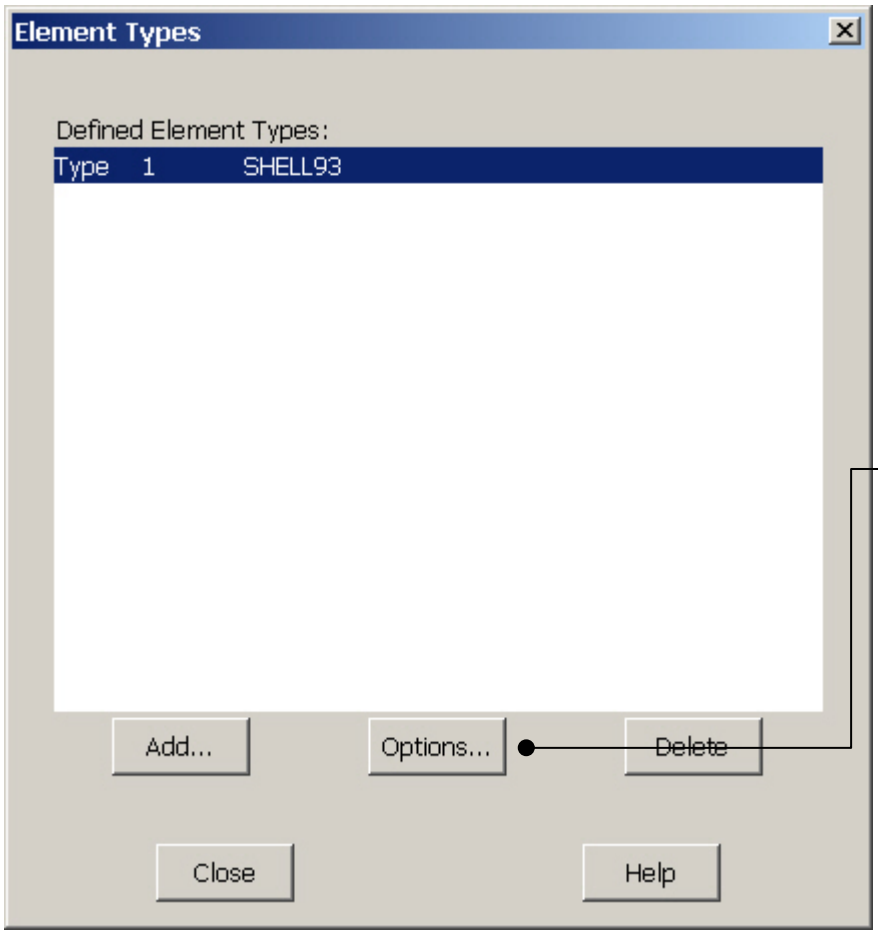
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

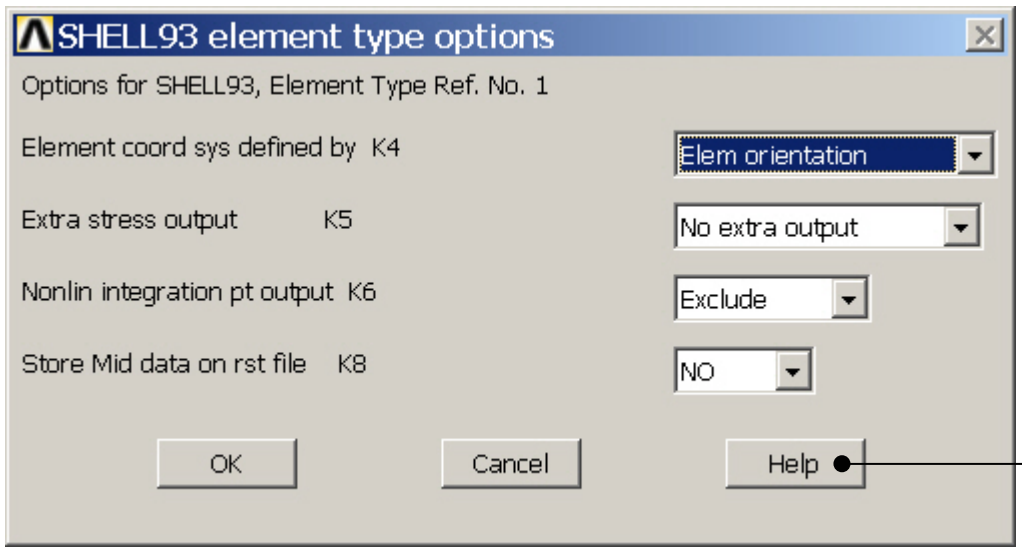
Preprocessor > Element Type > Add/Edit/Delete



Press Options

Example - Element Type

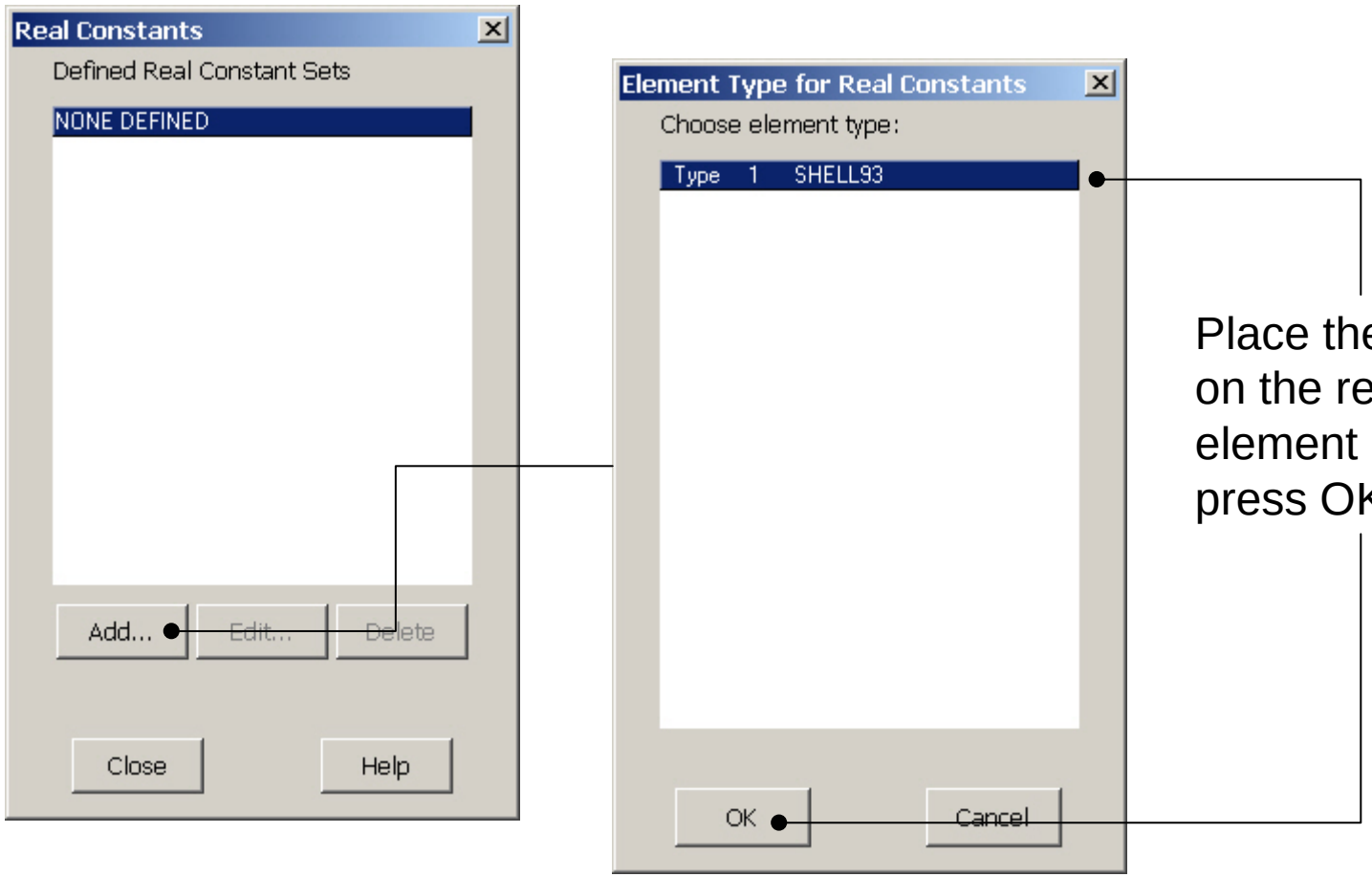
Preprocessor > Element Type > Add/Edit/Delete



Press Help to learn more about the element.

Example – Real Constants

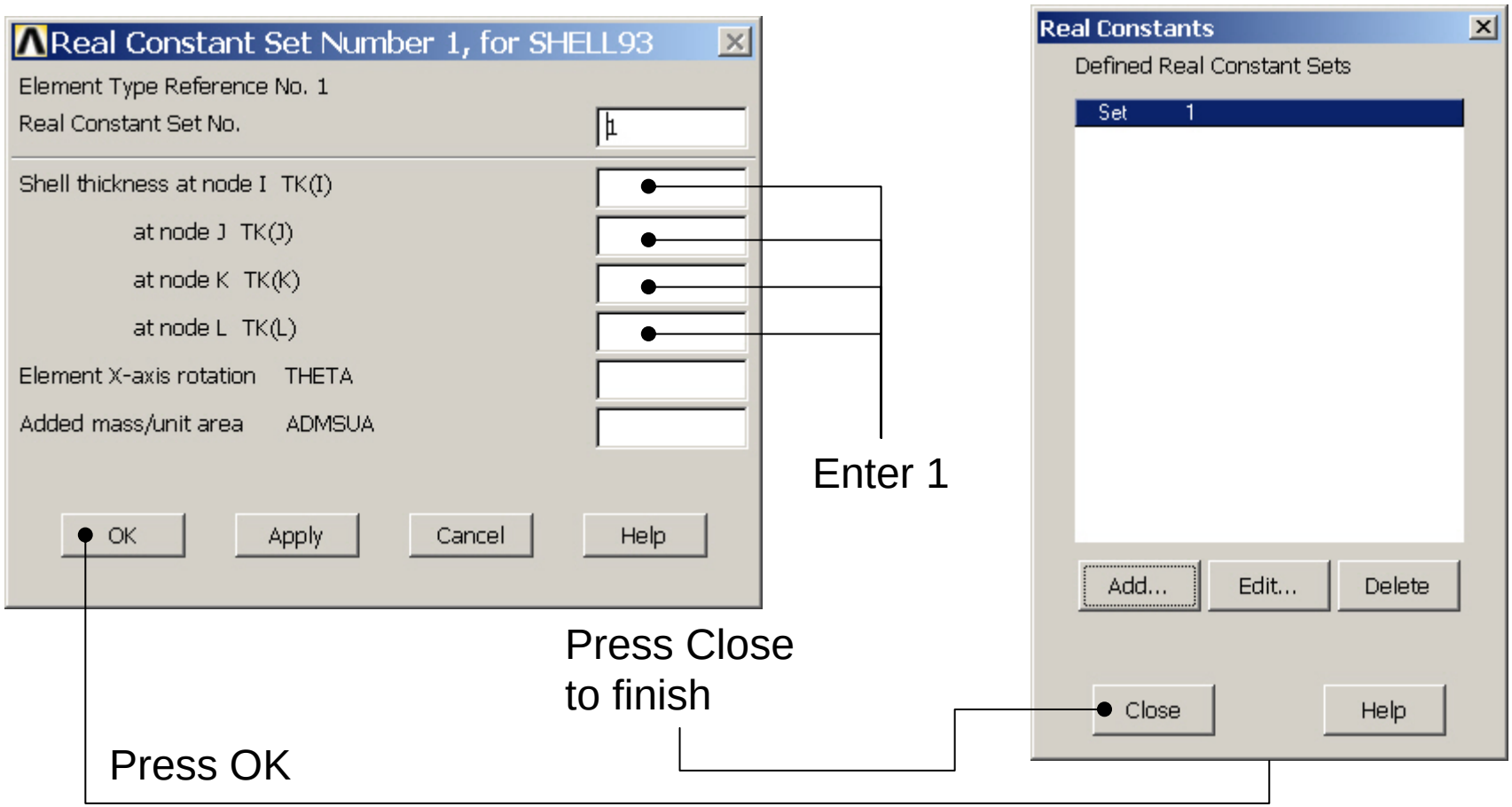
Preprocessor > Real Constants > Add



Place the cursor on the relevant element and press OK

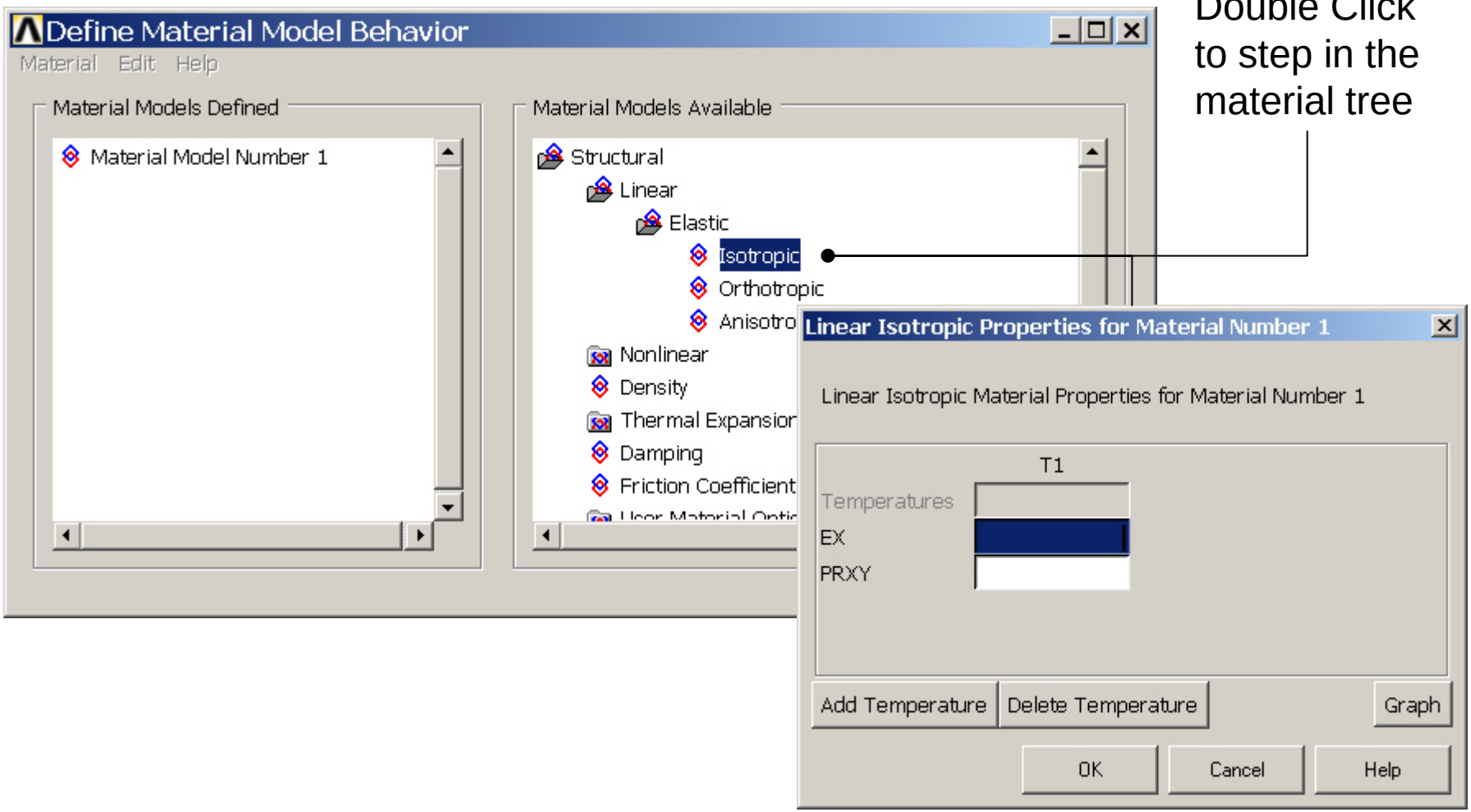
Example - Real Constants

Preprocessor > Real Constants > Add



Example - Material Properties

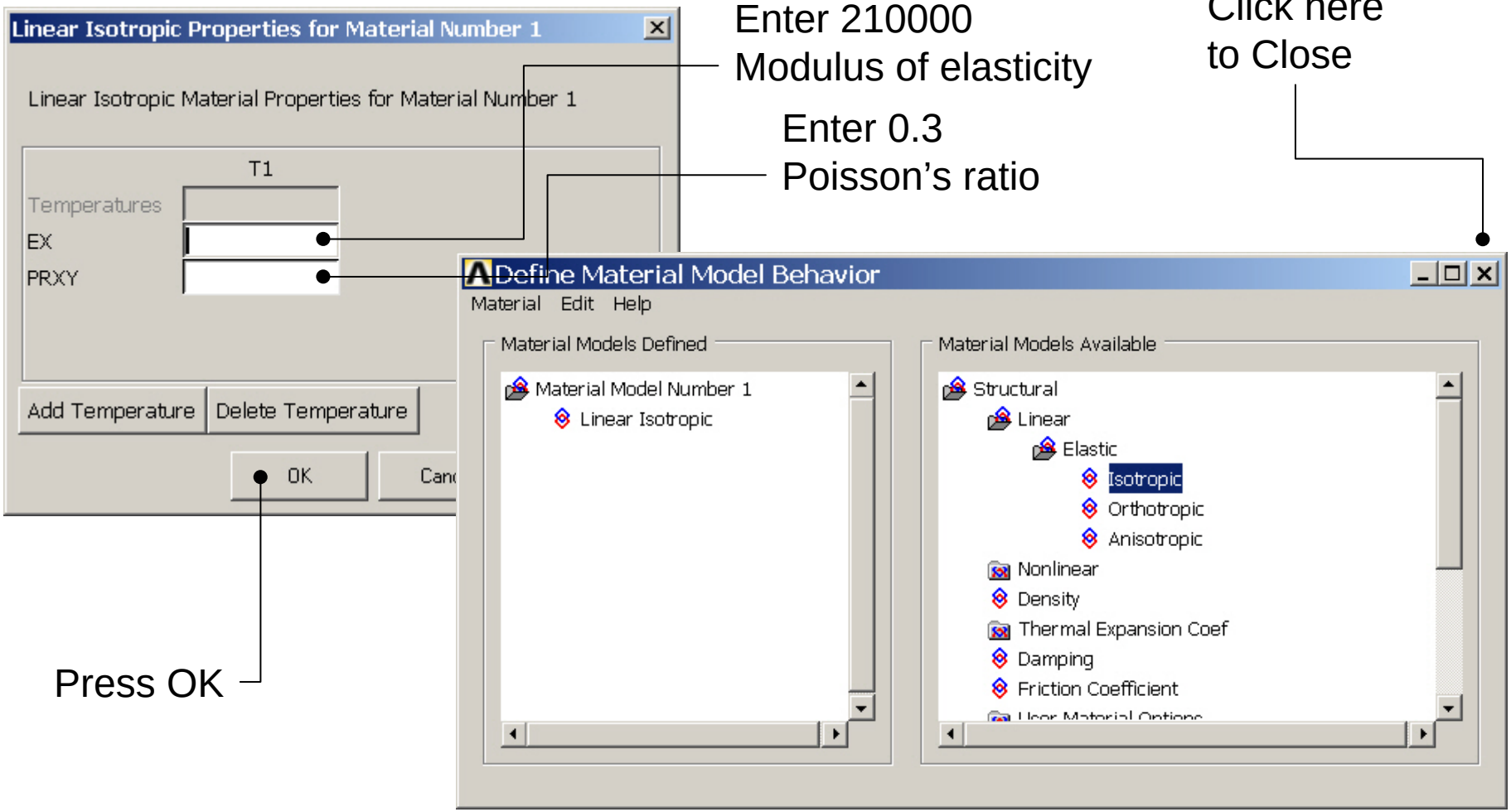
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

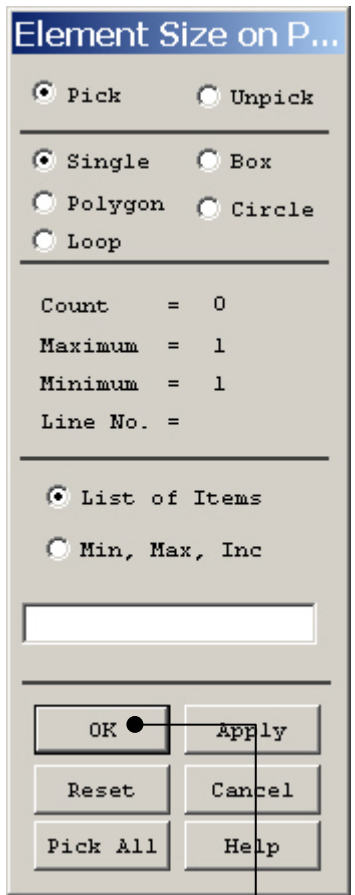
Example - Material Properties

Preprocessor > Material Props > Material Models

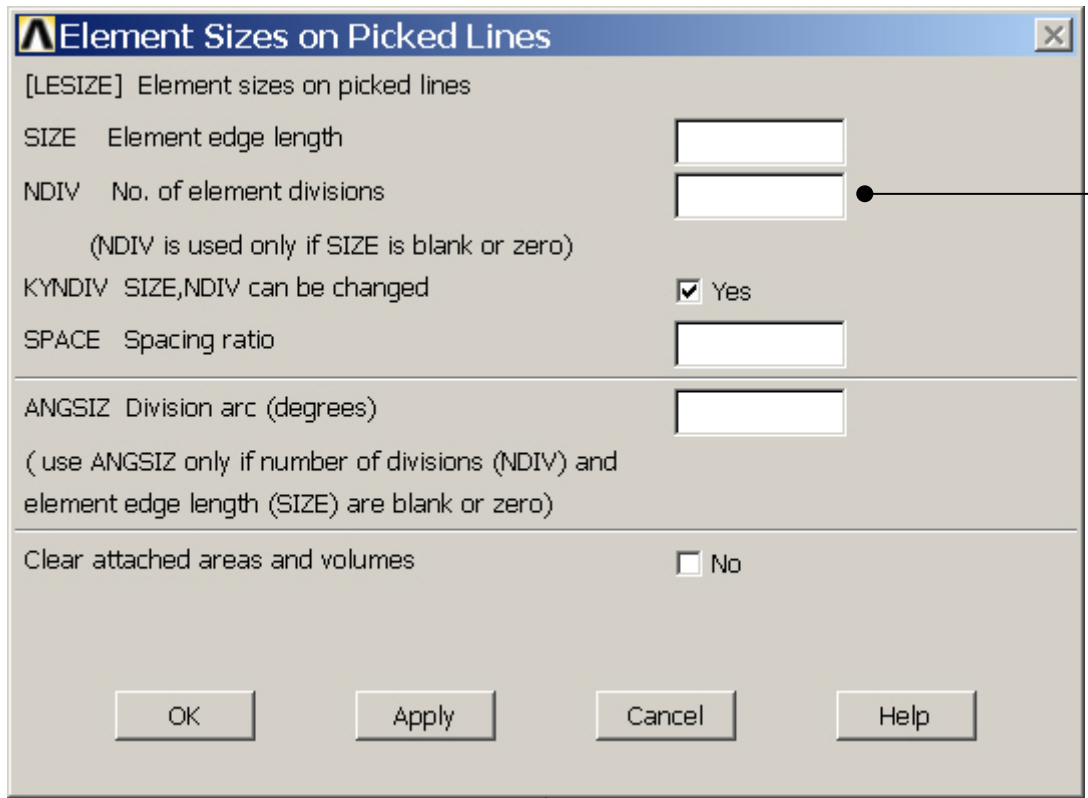


Example - Meshing

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



Select/Pick the two shortest lines to specify mesh size for

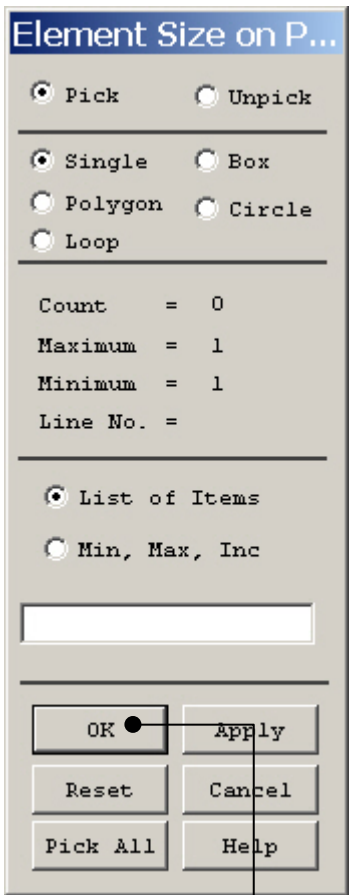


Press OK when finish with selection

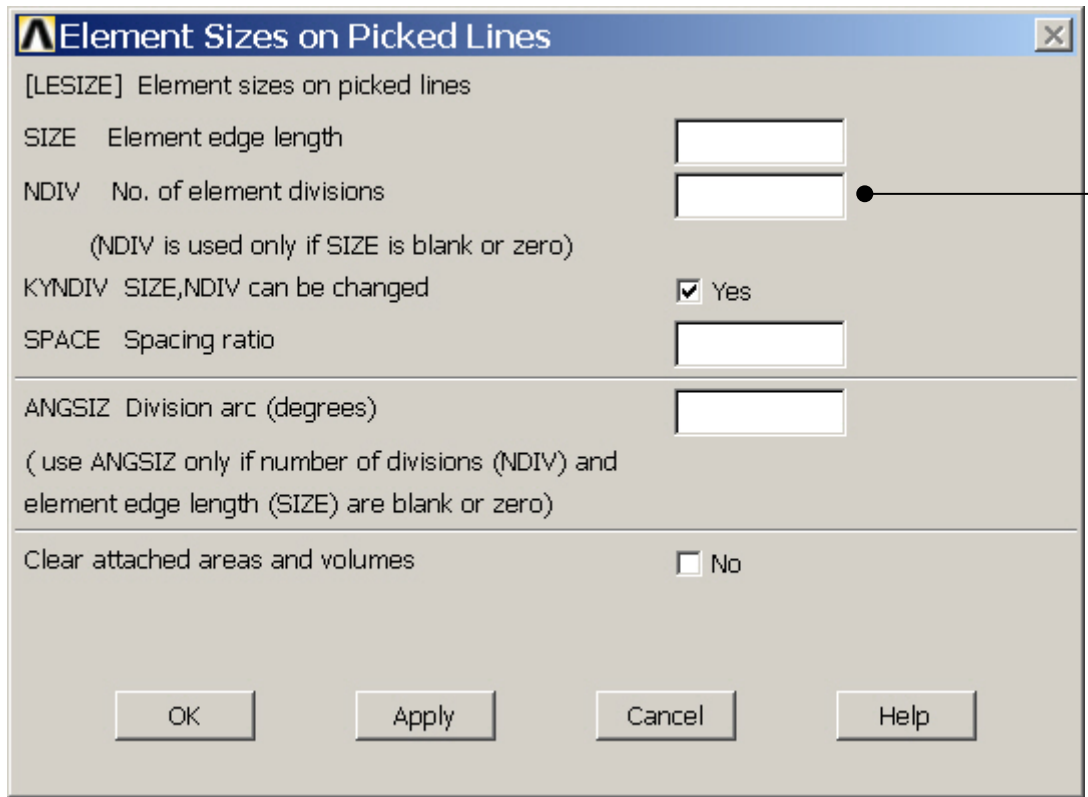
Enter 10

Example - Meshing

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



Select/Pick the two longest lines to specify mesh size for

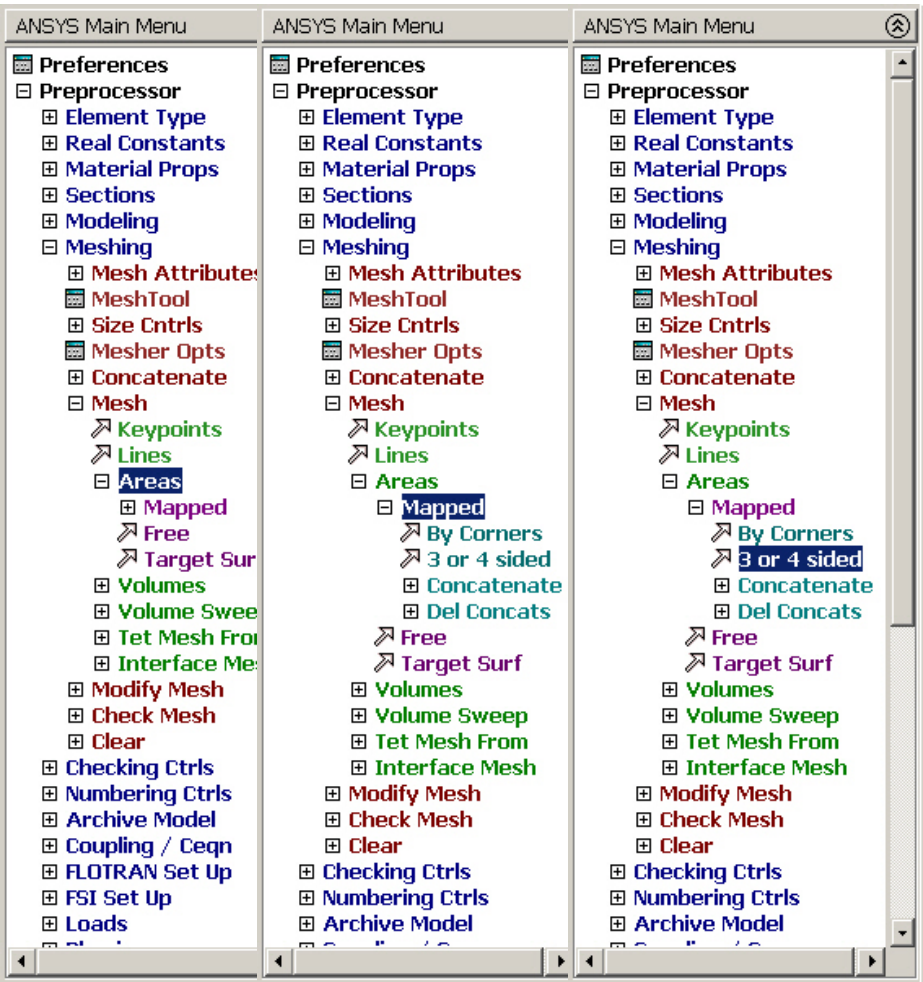


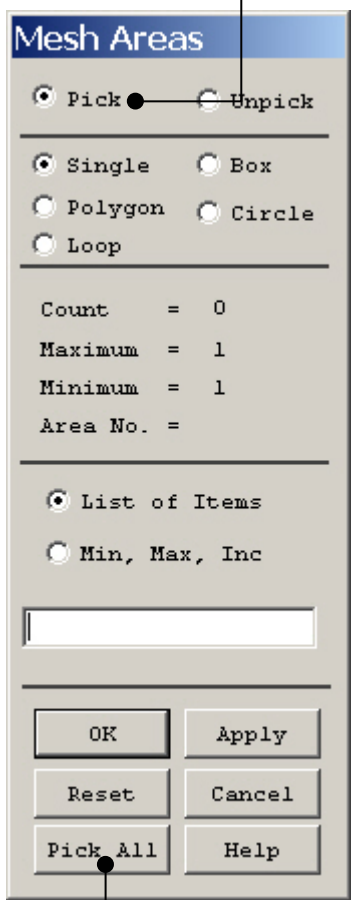
Press OK when finish with selection

Enter 20

Example - Meshing

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





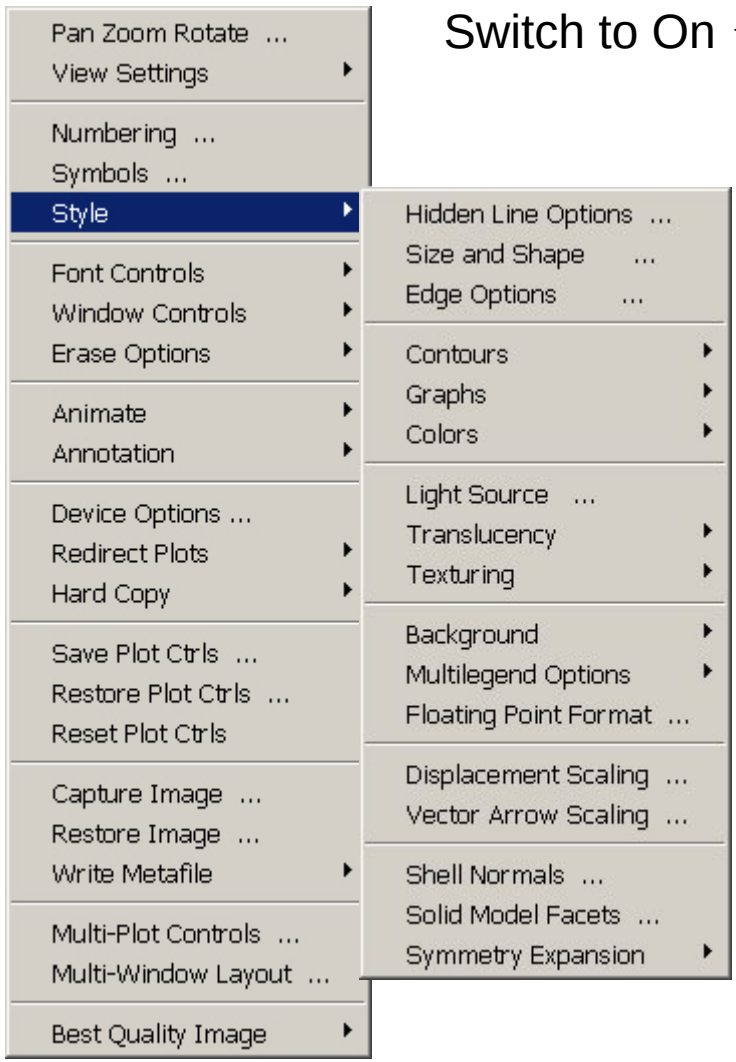
Select individual areas to be meshed

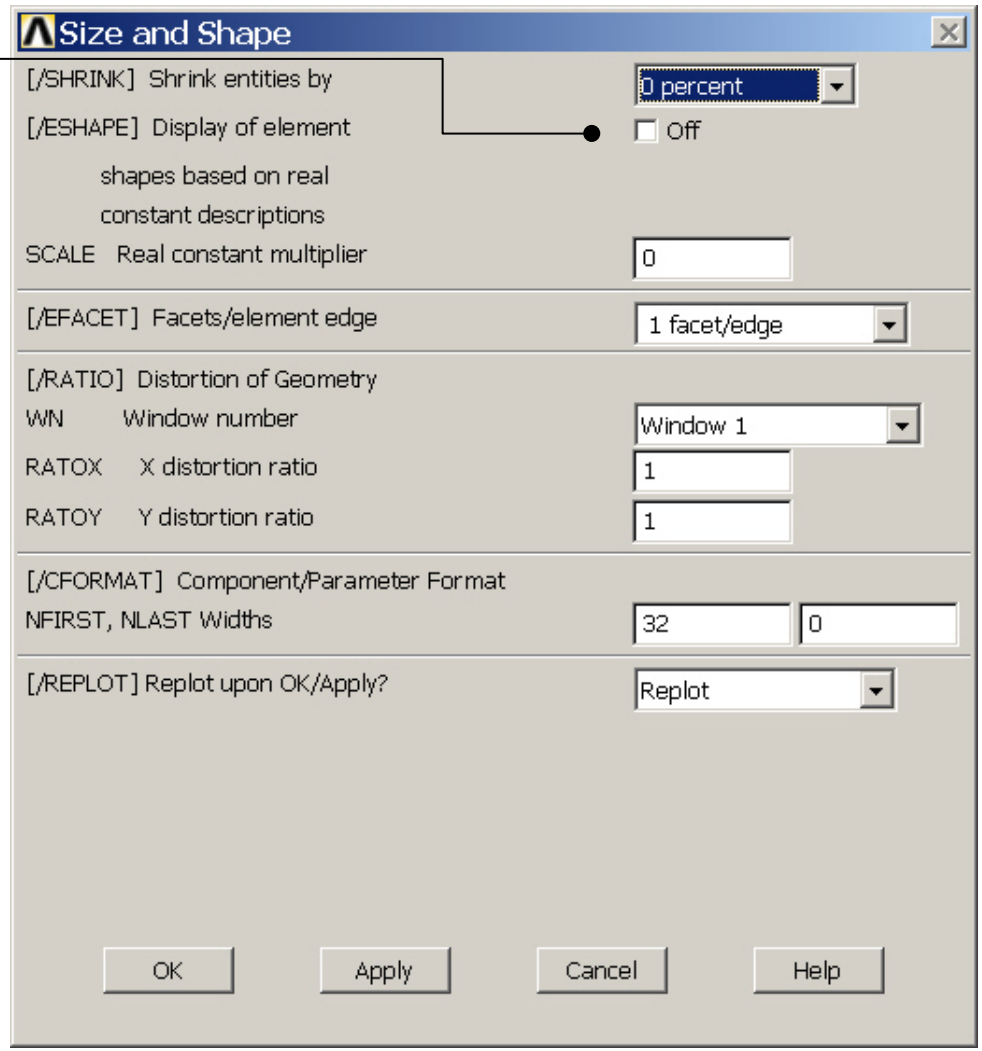
NB: It is often necessary to “Clear” the model for example if Element Type or model geometry is to be changed

Select all areas defined to be meshed

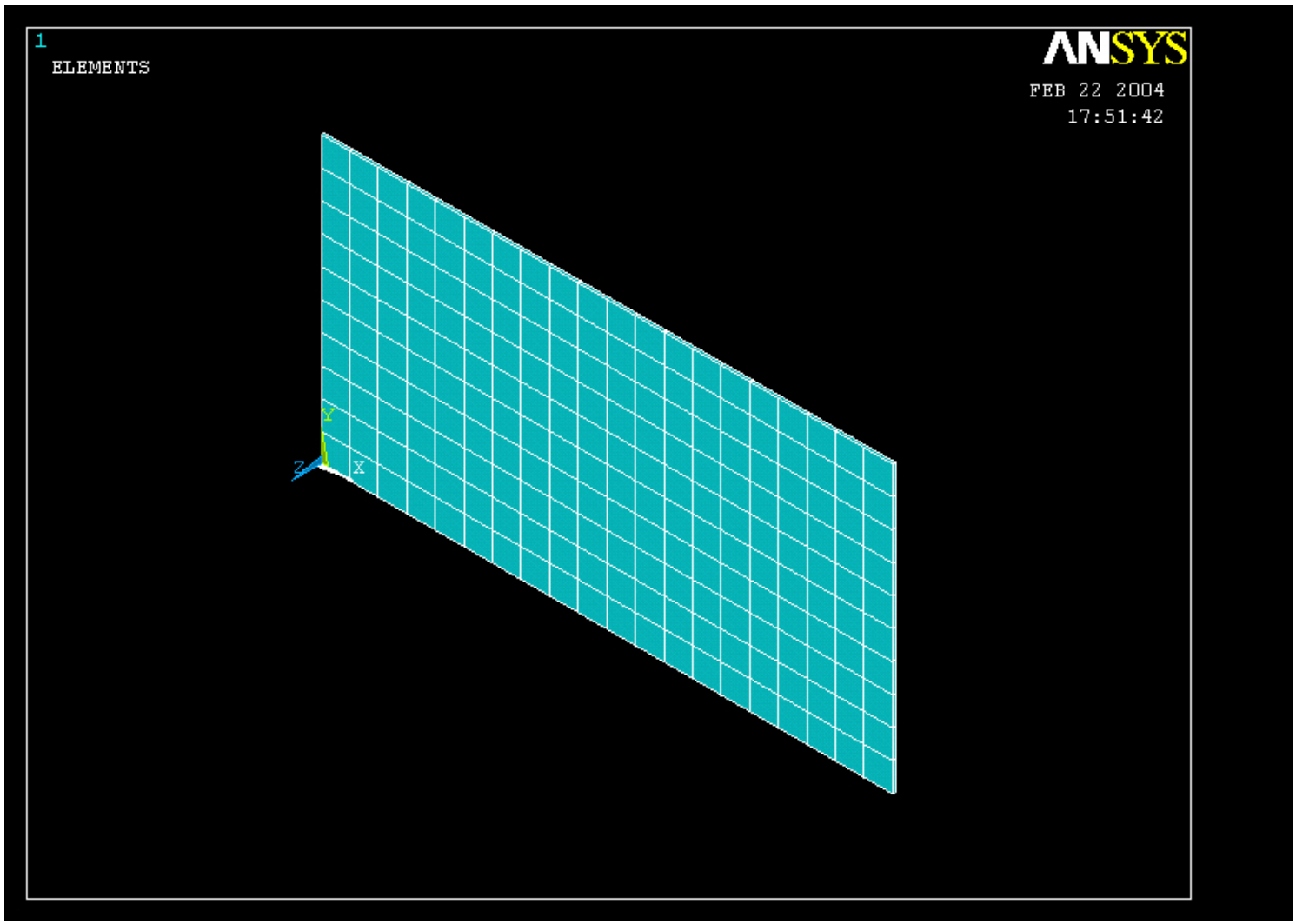
Example - PlotCtrls Menu

Switch to On





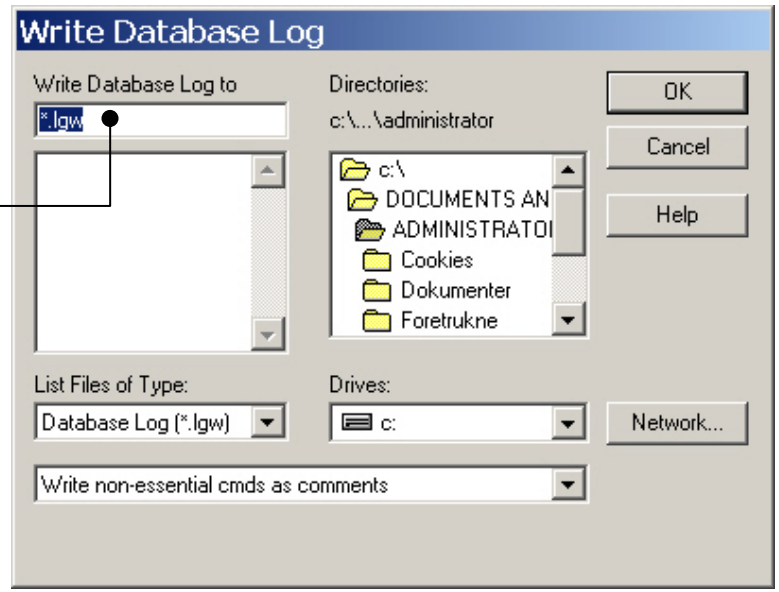
Example – Display of Element



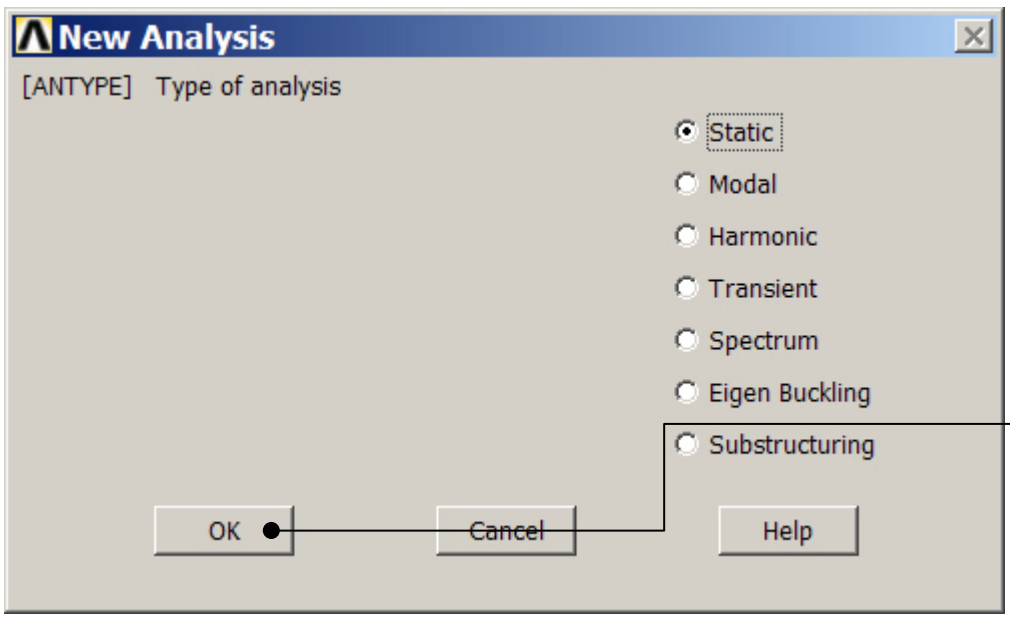
Example – Analysis Type

File > Write DB log file

Enter “example0505.lgw”

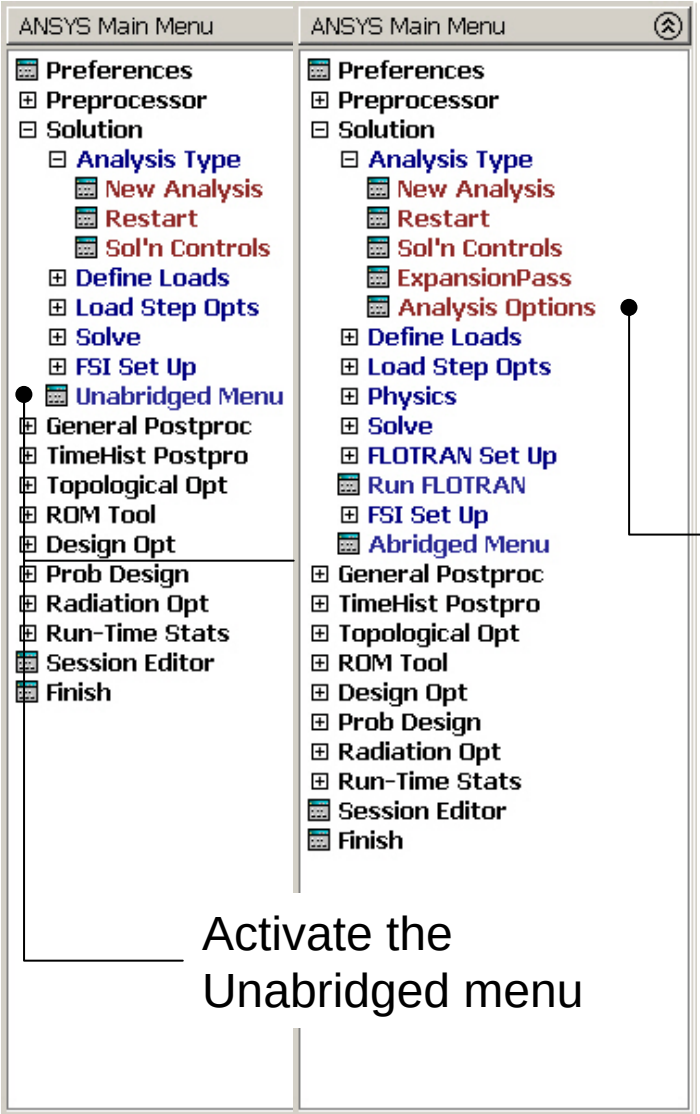


Solution > Analysis Type > New Analysis

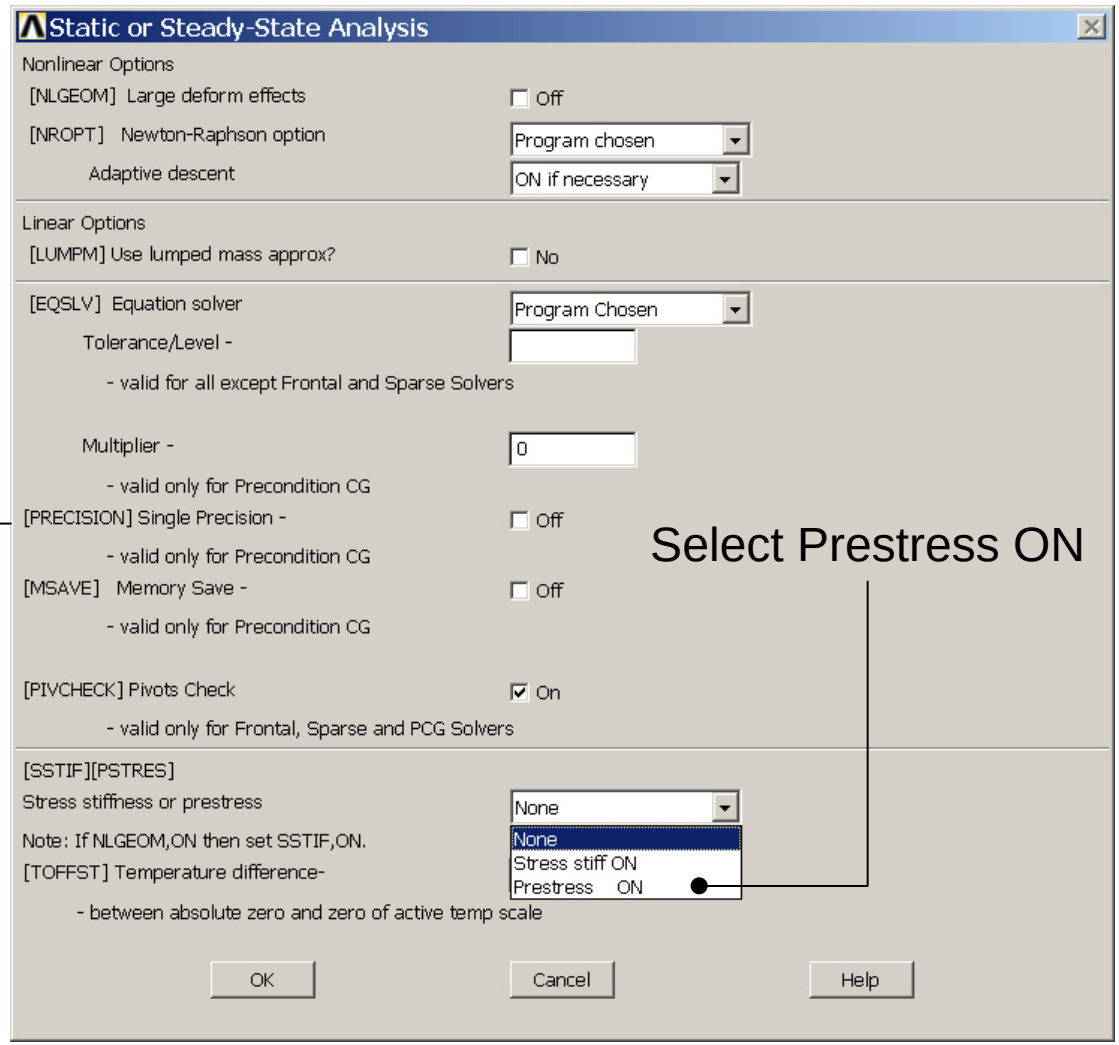


Press OK

Static solution – Analysis Options



Activate the Unabridged menu

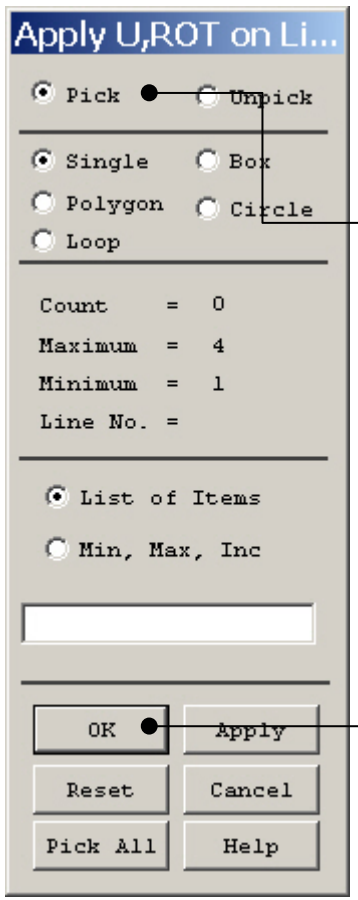


Select Prestress ON

Example0505

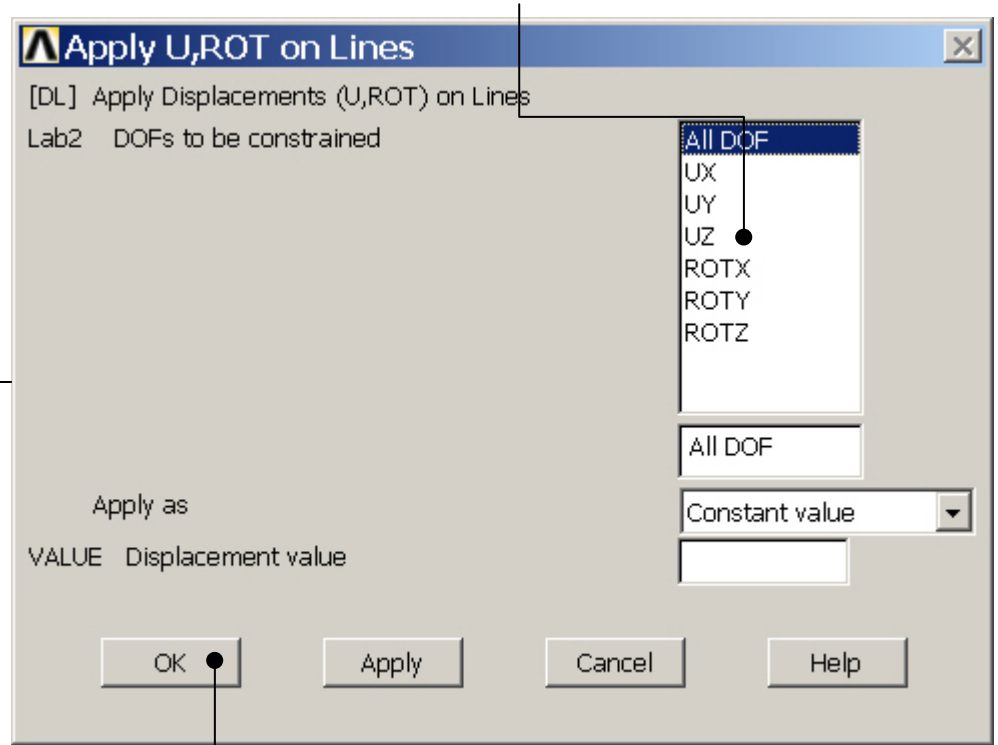
Example – Define Loads

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



Select the bottom straight line

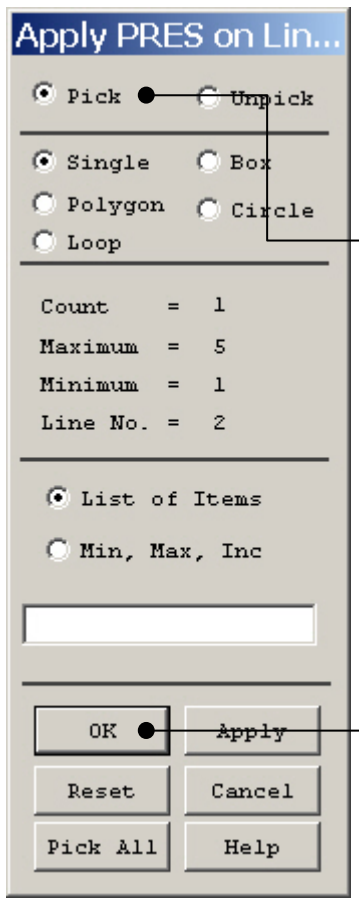
Select UZ to fix the plate in the z-direction



Press OK

Example – Define Loads

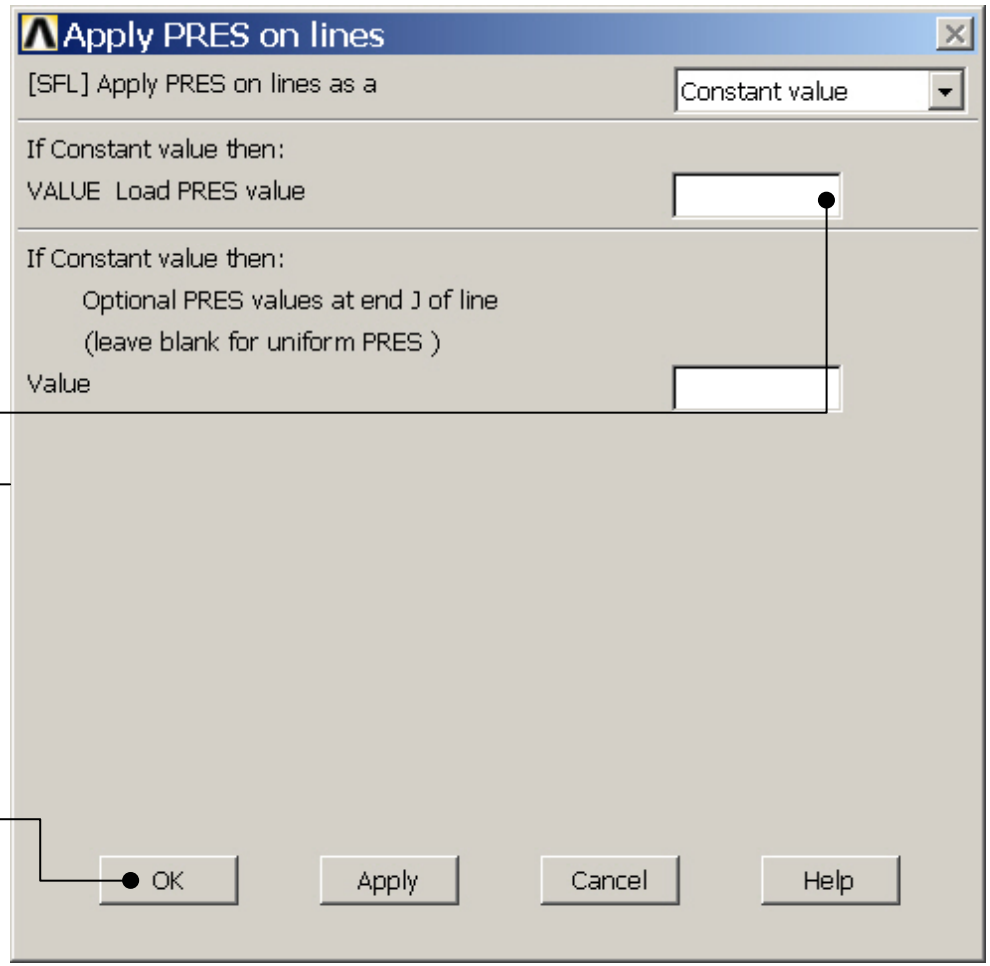
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the right straight line

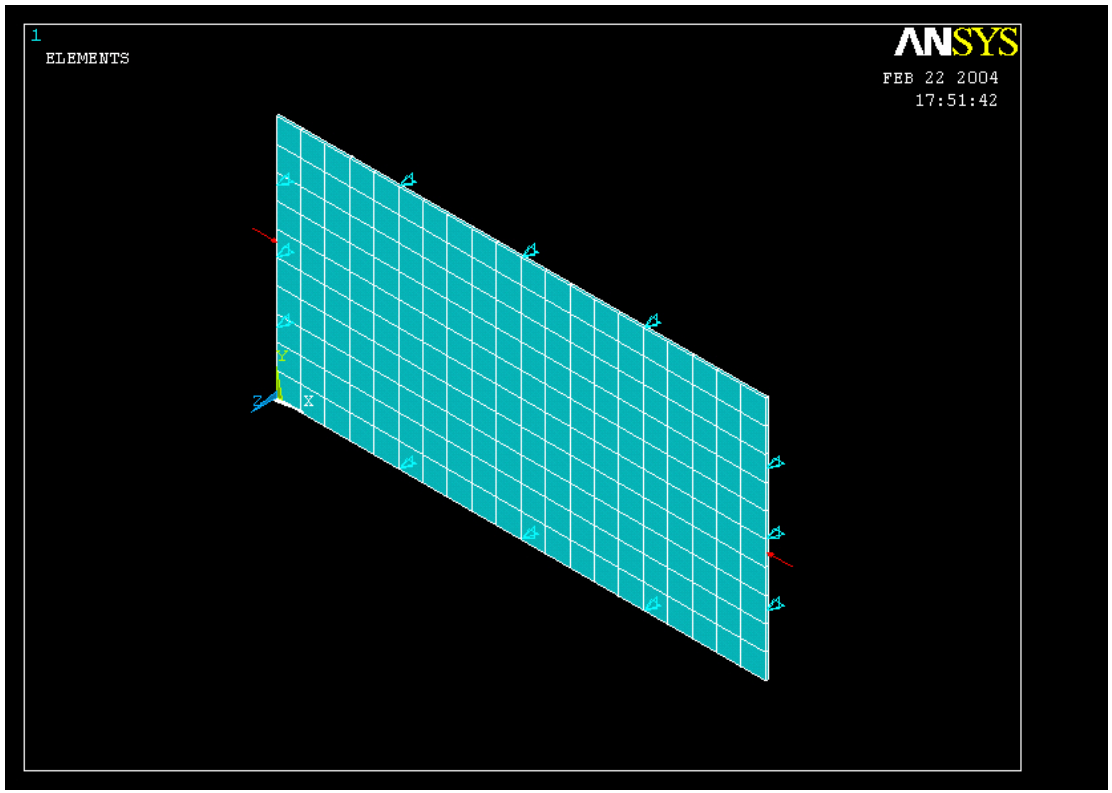
Enter -1

Press OK to finish

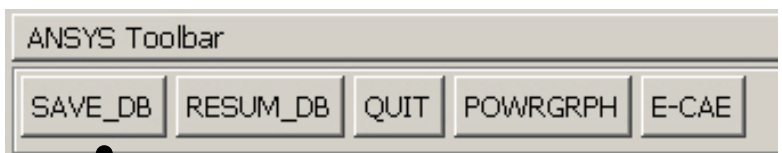


Note: Pressure acts normal and inward to a surface

Example - Save



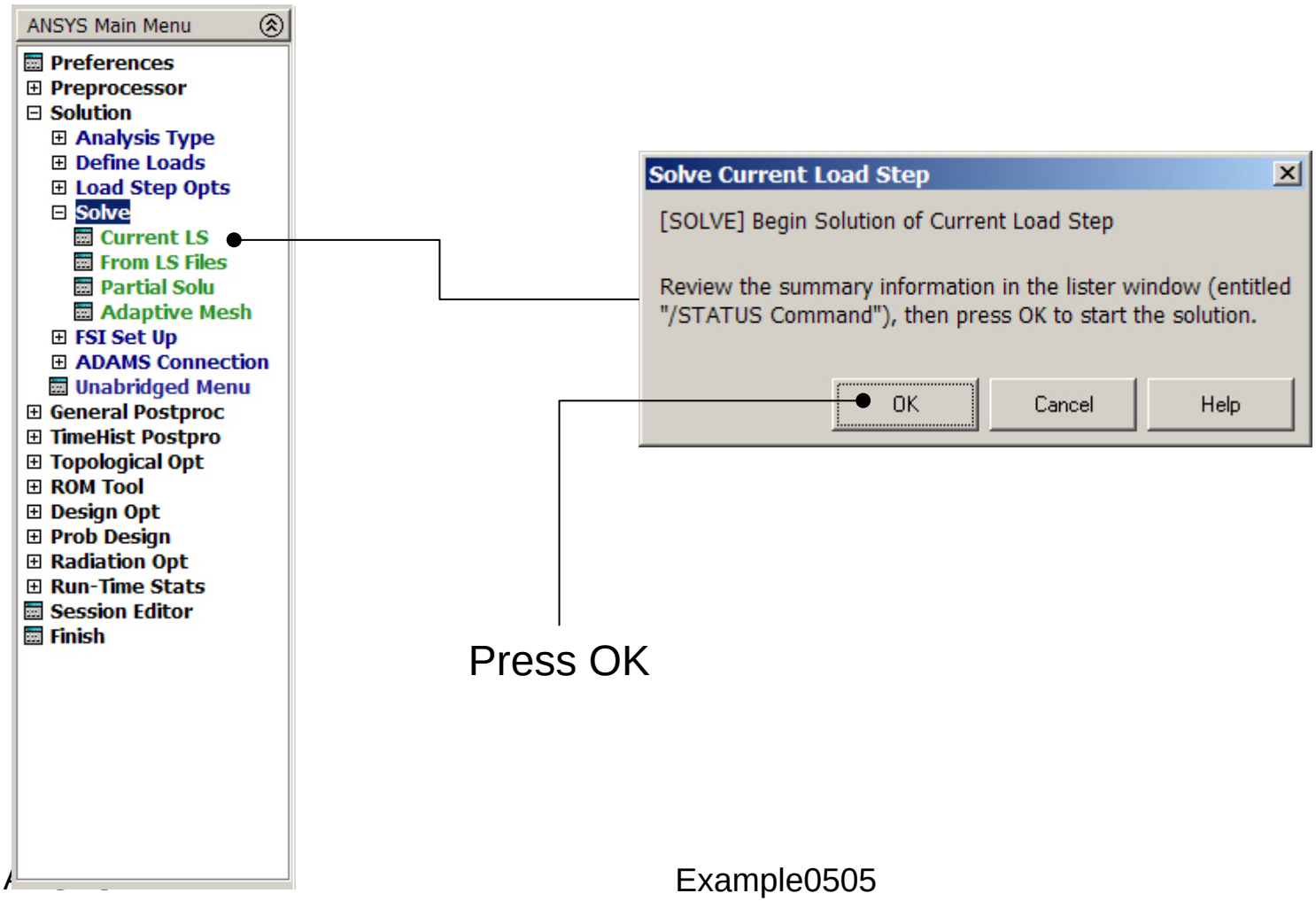
Display of Analysis model



Save the model

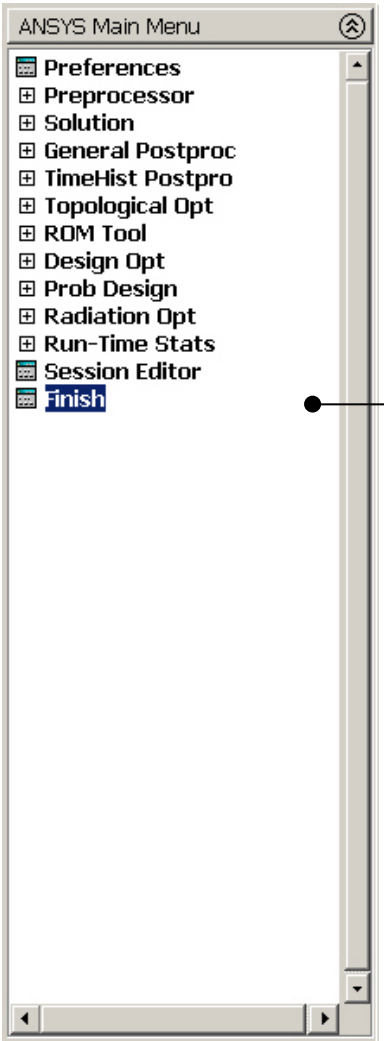
Example - Solve

Solution > Solve > Current LS



Press OK

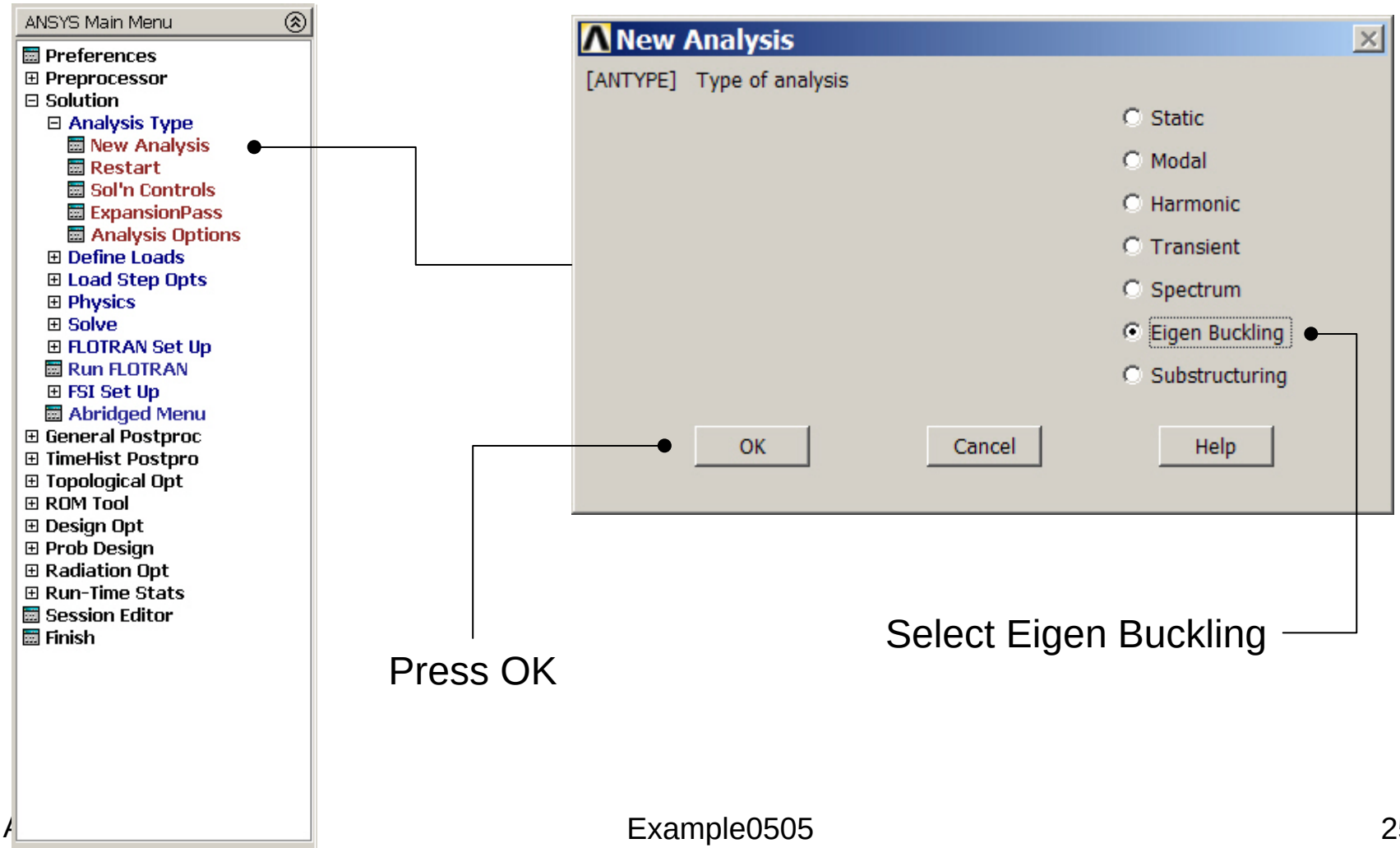
Example - Finish



Press Finish to end the static solution

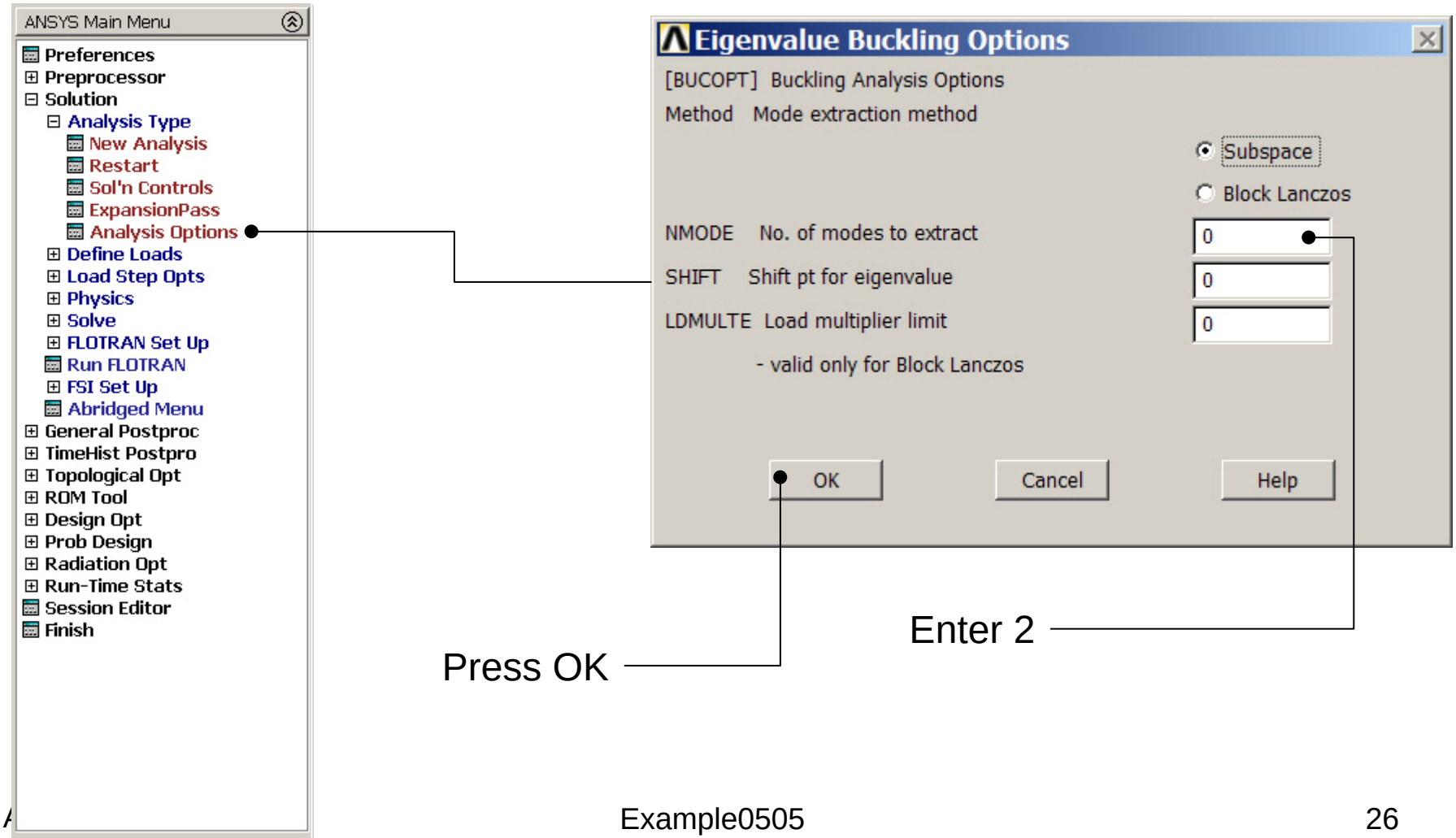
Eigen Buckling - New Analysis

Main Menu> Solution> Analysis Type> New Analysis

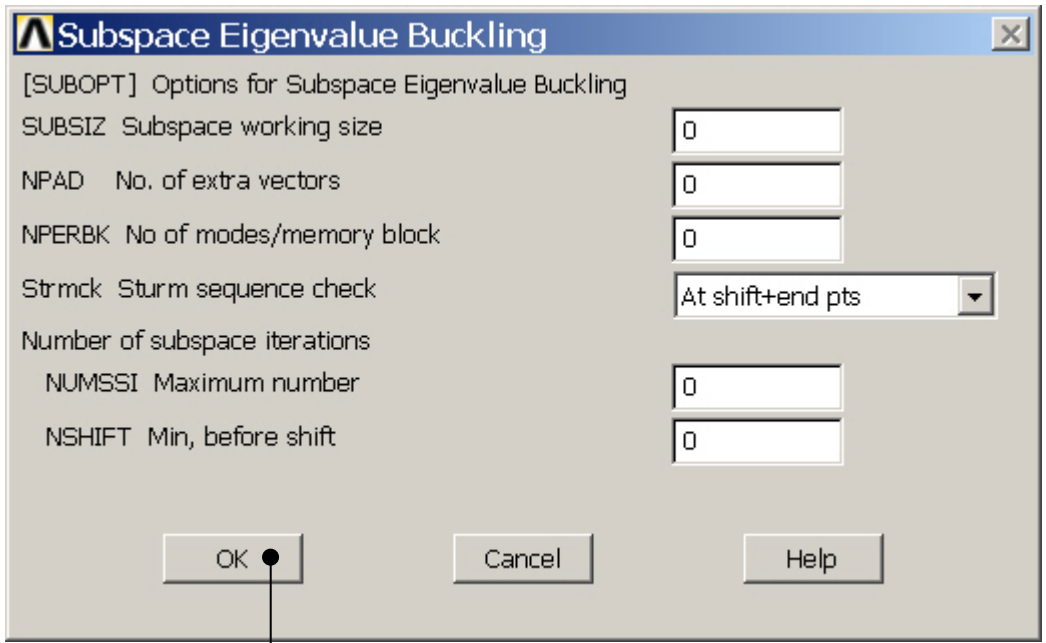


Eigen Buckling – Analysis Options

Main Menu> Solution> Analysis Type> Analysis Options



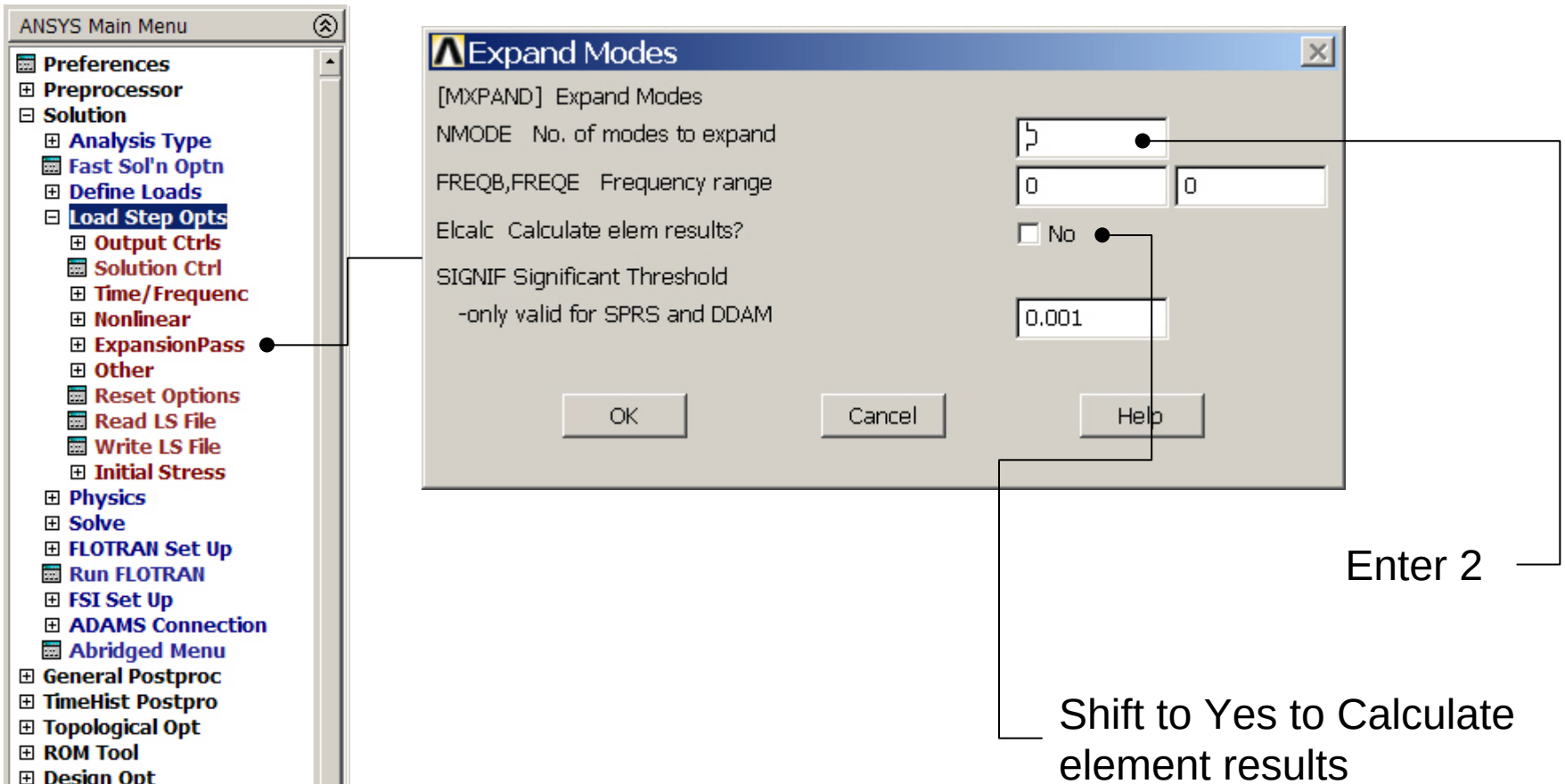
Example – Subspace Options



Press OK

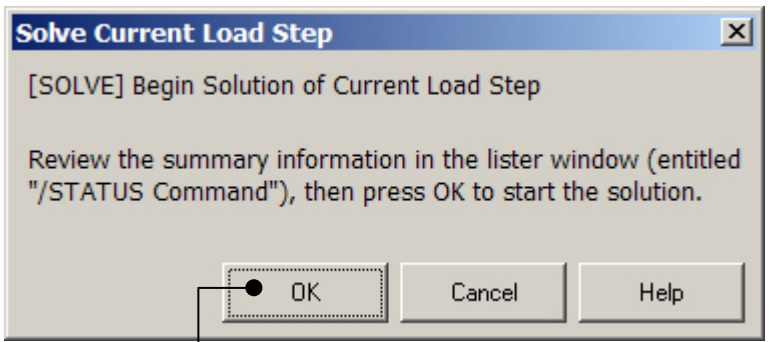
Eigen Buckling – Expanding Modes

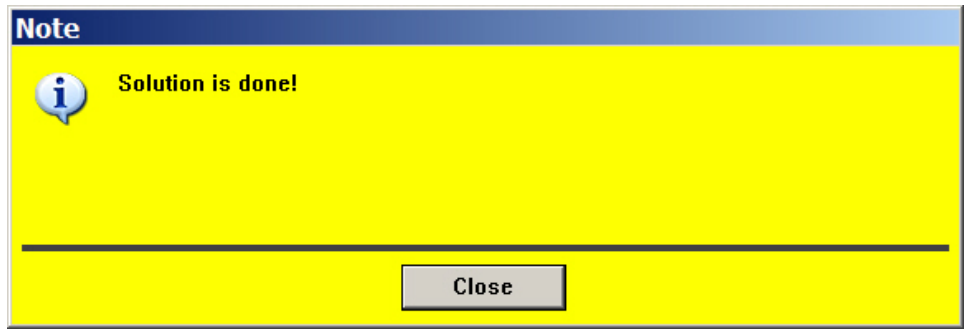
Main Menu> Solution> Load Step Opts > ExpansionPass > Single Expand > Expand Modes



Example - Solve

Solution > Solve > Current LS





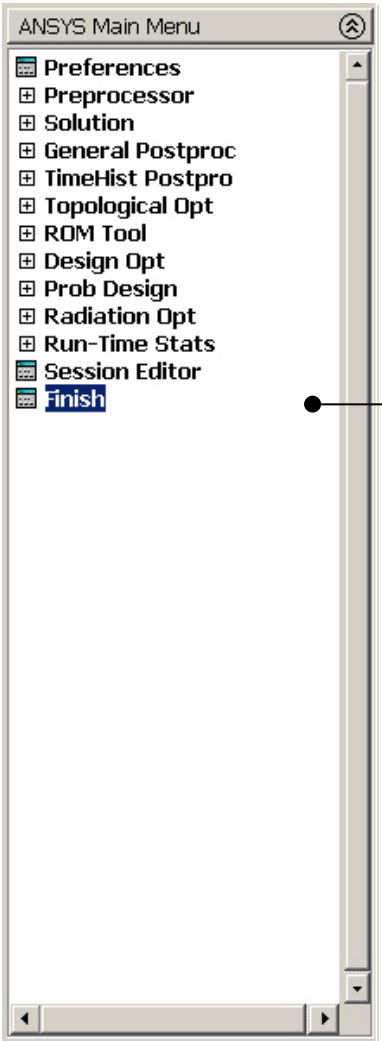
Press OK

Example – Output Window

ANSYS 7.1 Output Window

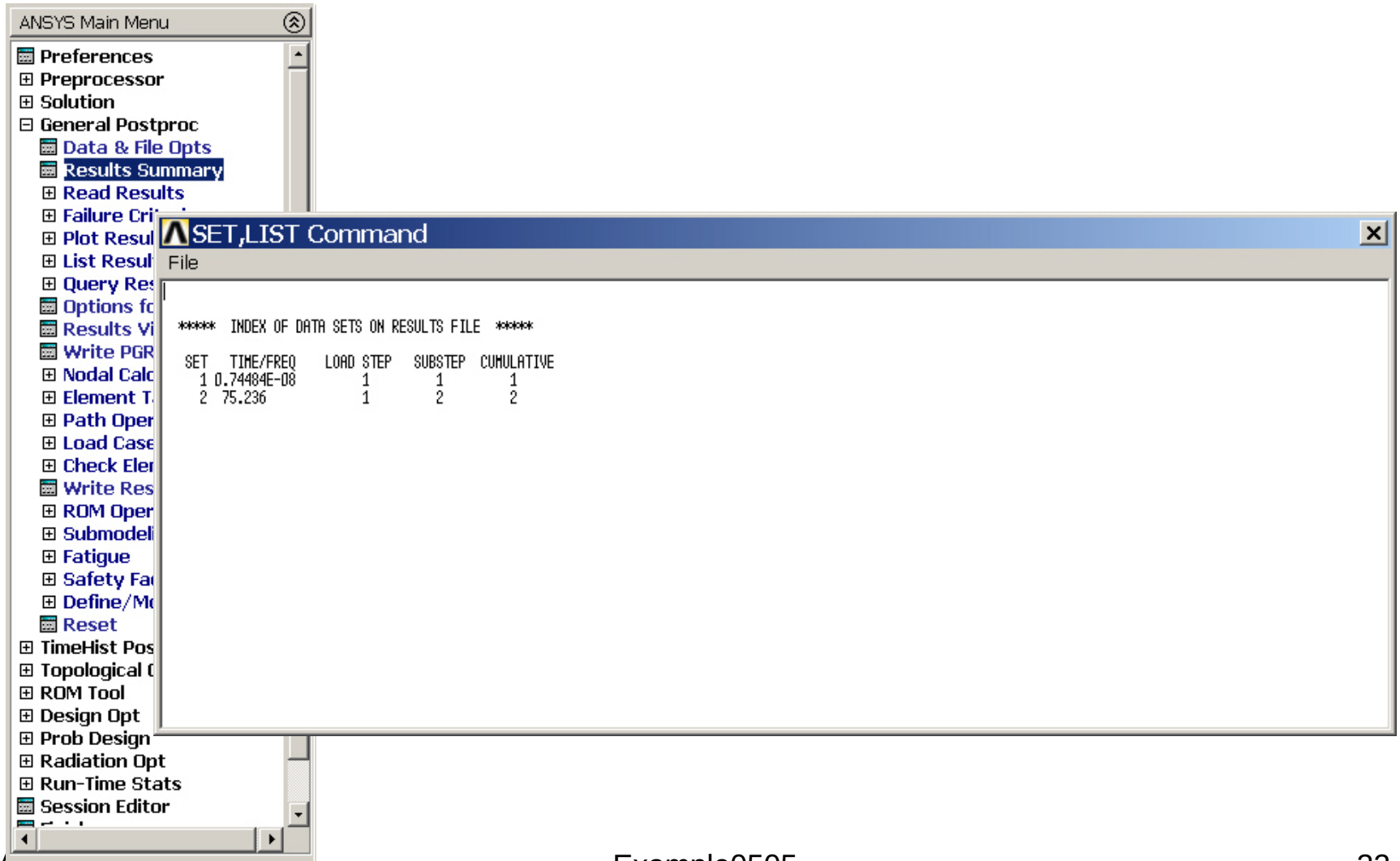
```
              *** FROM SUBSPACE ITERATION ***

     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 ***                            CP=     24.125   TIME= 18:04
Solution is done!


  *** 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

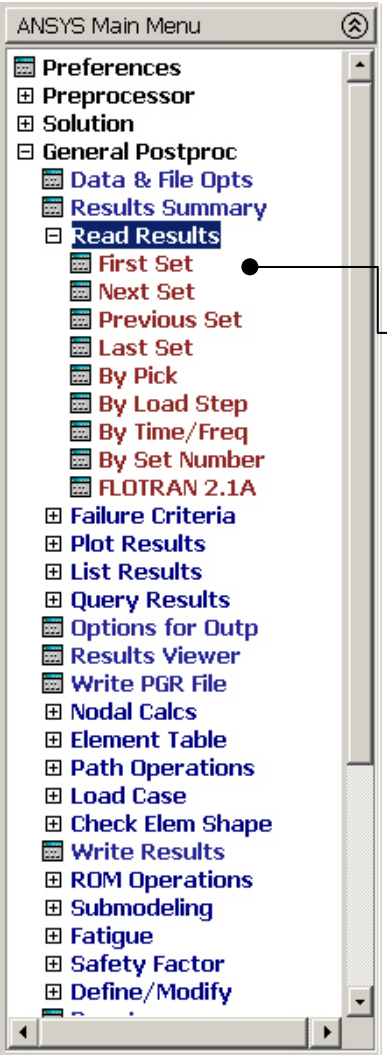


Press Finish to end the eigen buckling solution

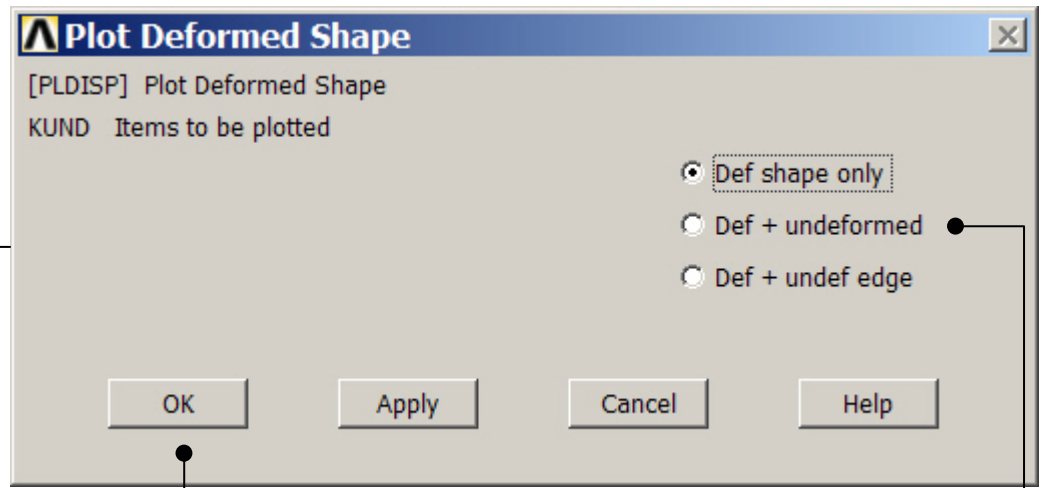
Example – Results Summary



Example – Read Results



General Postproc > Plot Results > Deformed Shape

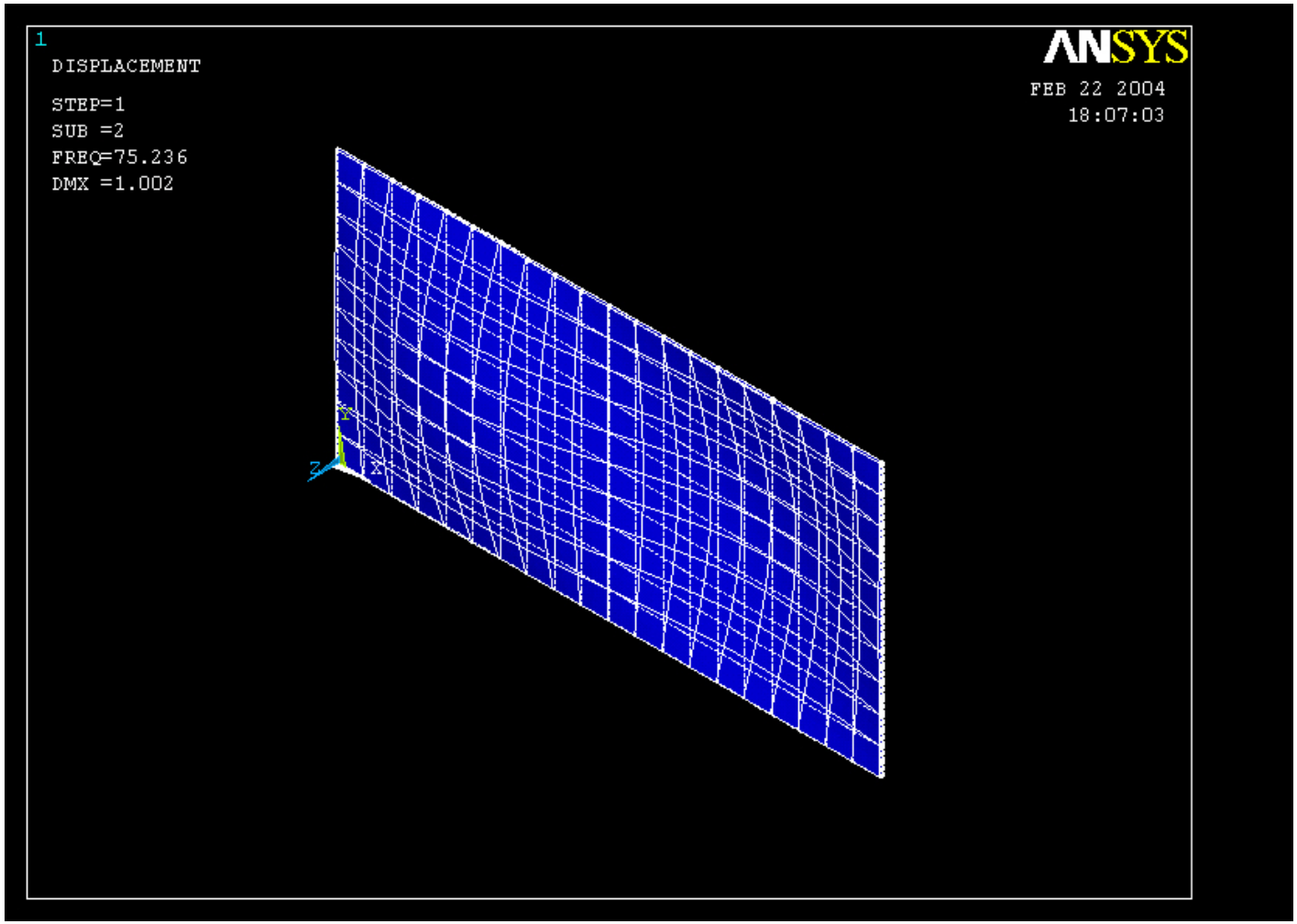


Select “Def+undeformed” and Press OK

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