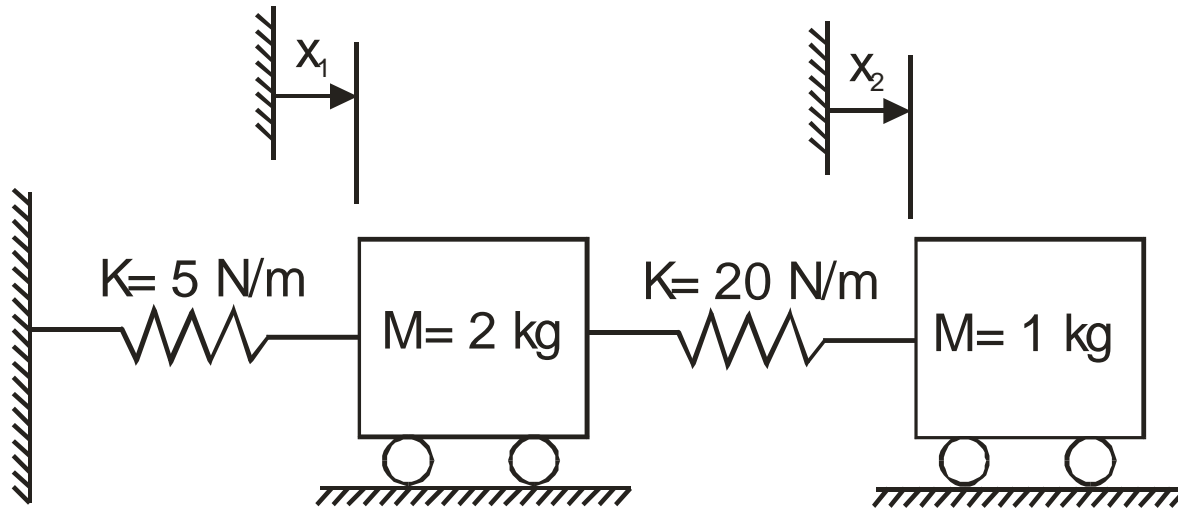


# Course in ANSYS

Example0430

# Example – Spring mass system



## Objective:

Compute the eigen frequency modes of the system

## Tasks:

Display the eigen modes of the system?

## Topics:

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

$$E = 1 \text{ N/m}^2$$

$$\nu = 0$$

$F = \text{none applied}$   
for modal analysis

$$x_1 = 5$$

$$x_2 = 11$$

# Example - title

**Utility Menu > File > Change Jobname**

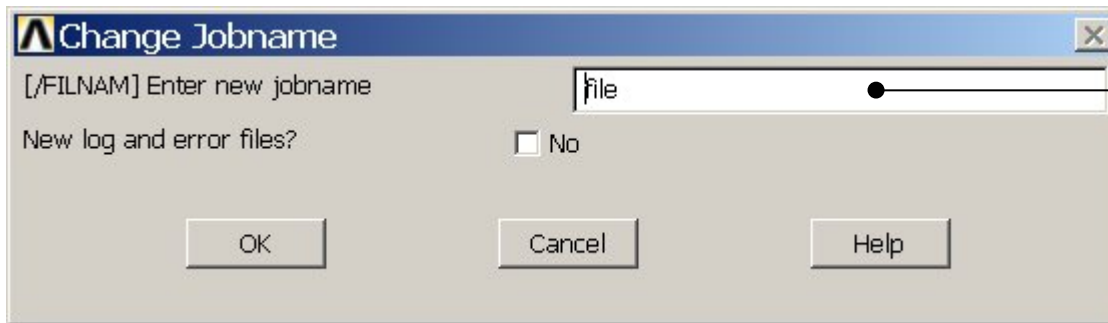


GUI

/jobname, Example0430



Command line entry

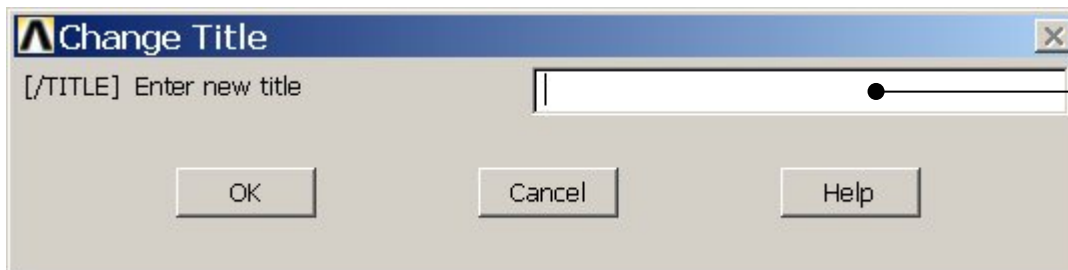


Enter: Example0430

**Utility Menu > File > Change Title**

/title, Spring mass system

Enter: Spring mass system



# Example – Areas Rectangle

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

Create an area given by  $X=(0,100)$  and  $Y=(0,50)$

The screenshot 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 fields. Arrows point from text labels to these fields: 'Enter 5' points to the top-right field, 'Enter 6' points to the top-left field, 'Enter 0 or leave empty' points to the bottom-left field, and 'Enter 1' points to the bottom-right field. 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.

Enter 5

Enter 6

Enter 0 or leave empty

Enter 1

Press OK

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

Example0430

Computational mechanics, AAU, Esbjerg

# Example – Areas Rectangle

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

Create an area given by  $X=(0,100)$  and  $Y=(0,50)$

The screenshot 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 fields. Arrows point from text labels to specific parts of the dialog box: 'Enter 11' points to the top-right input field (X2), 'Enter 12' points to the top-left input field (X1), 'Enter 0 or leave empty' points to the bottom-left input field (Y1), and 'Enter 1' points to the bottom-right input field (Y2). Below the input fields are four buttons: 'OK', 'Apply', 'Cancel', and 'Help'. An arrow points from the text 'Press OK' to the 'OK' button.

Enter 11

Enter 12

Enter 0 or leave empty

Enter 1

Press OK

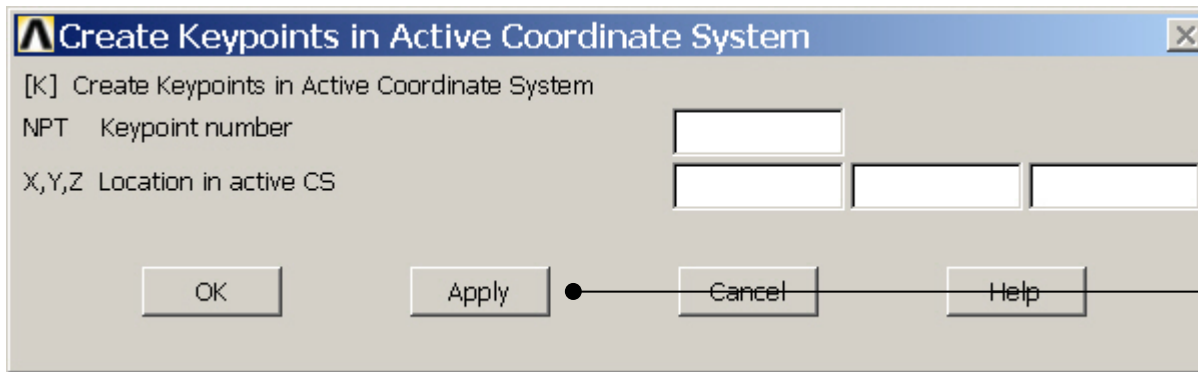
Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

Example0430

Computational mechanics, AAU, Esbjerg

# Example - Keypoints

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



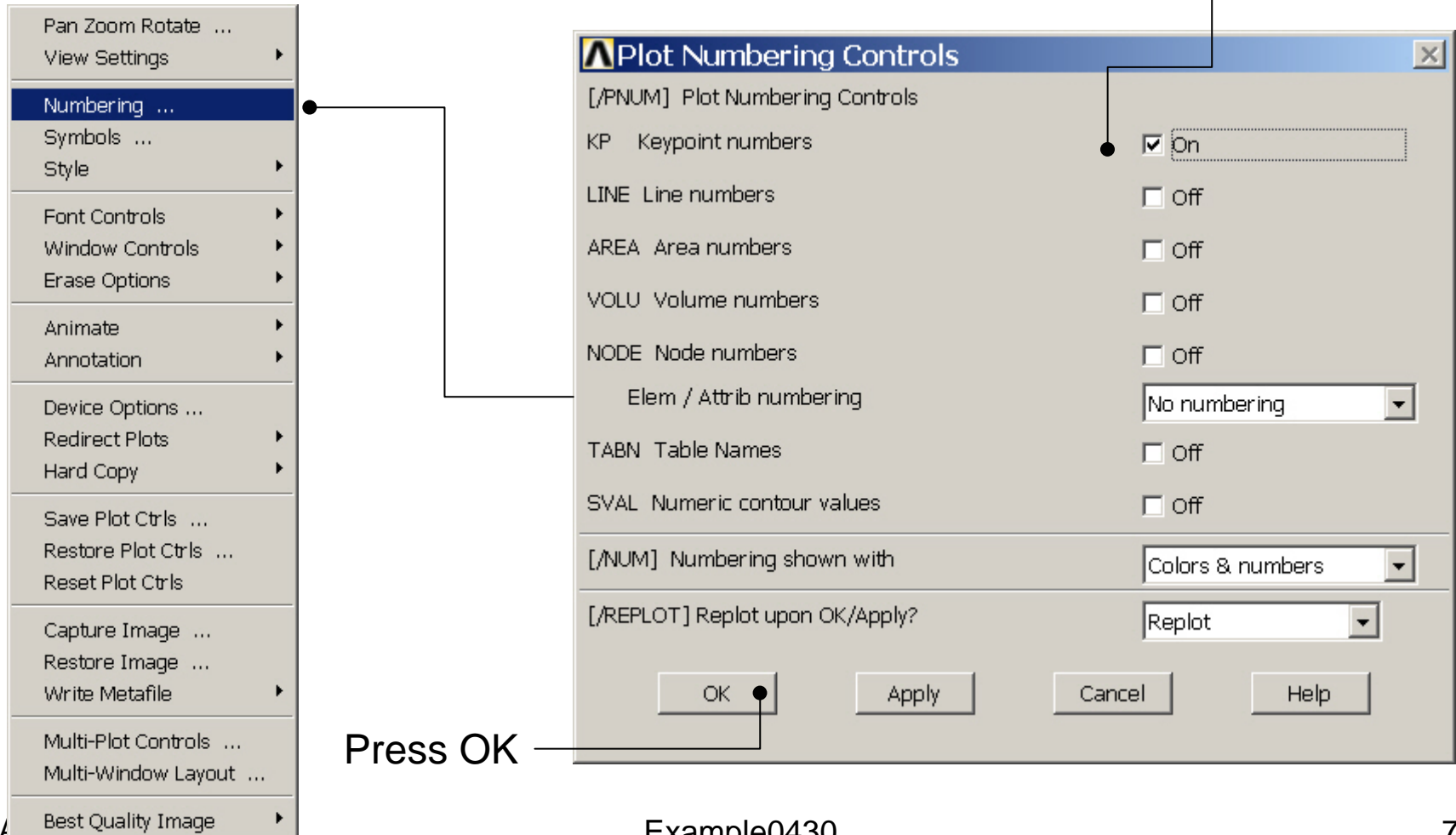
Enter 0,0.5,0 and  
Press **Apply**

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

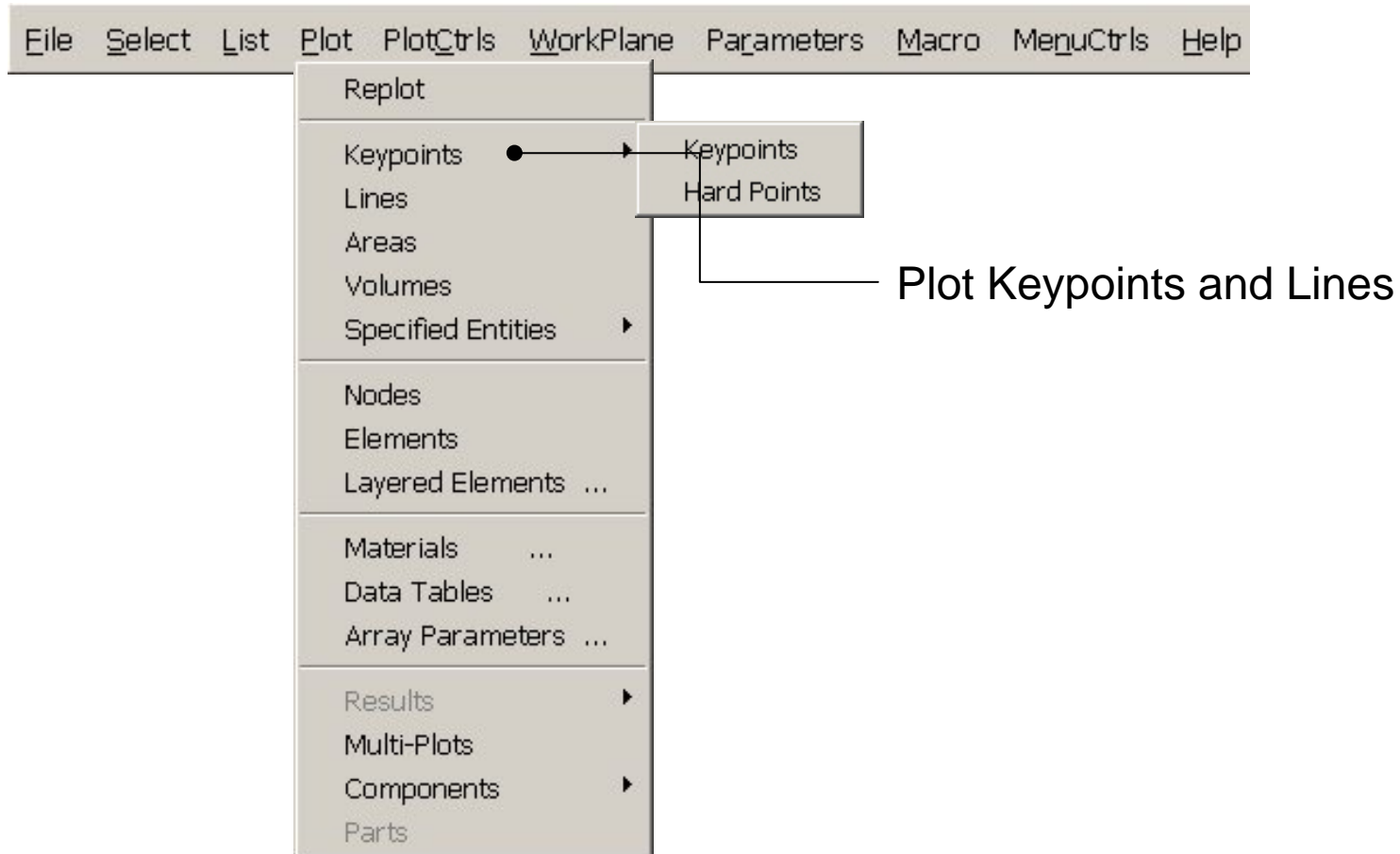
# Example - Numbering

Utility Menu > PlotCtrls > Numbering

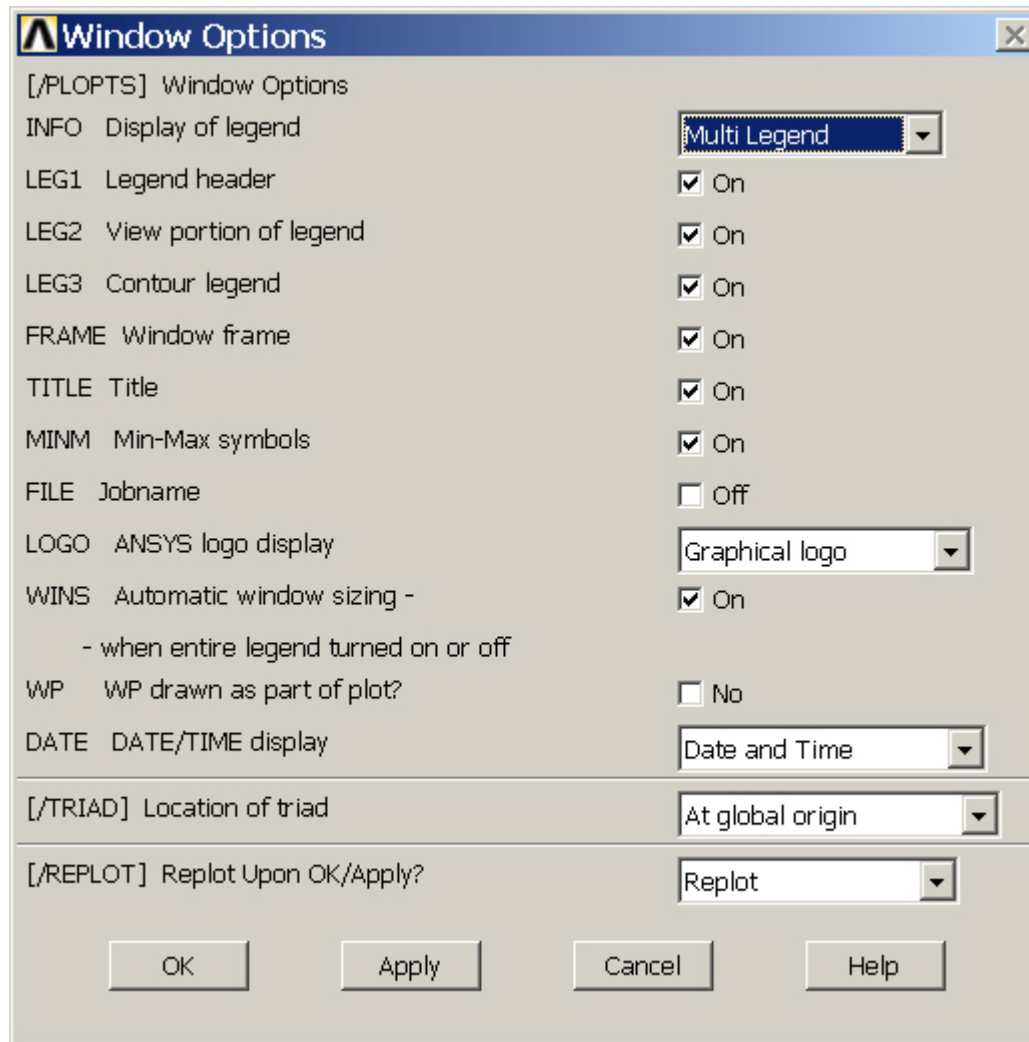
Switch on Keypoint numbers



# Example - Plot

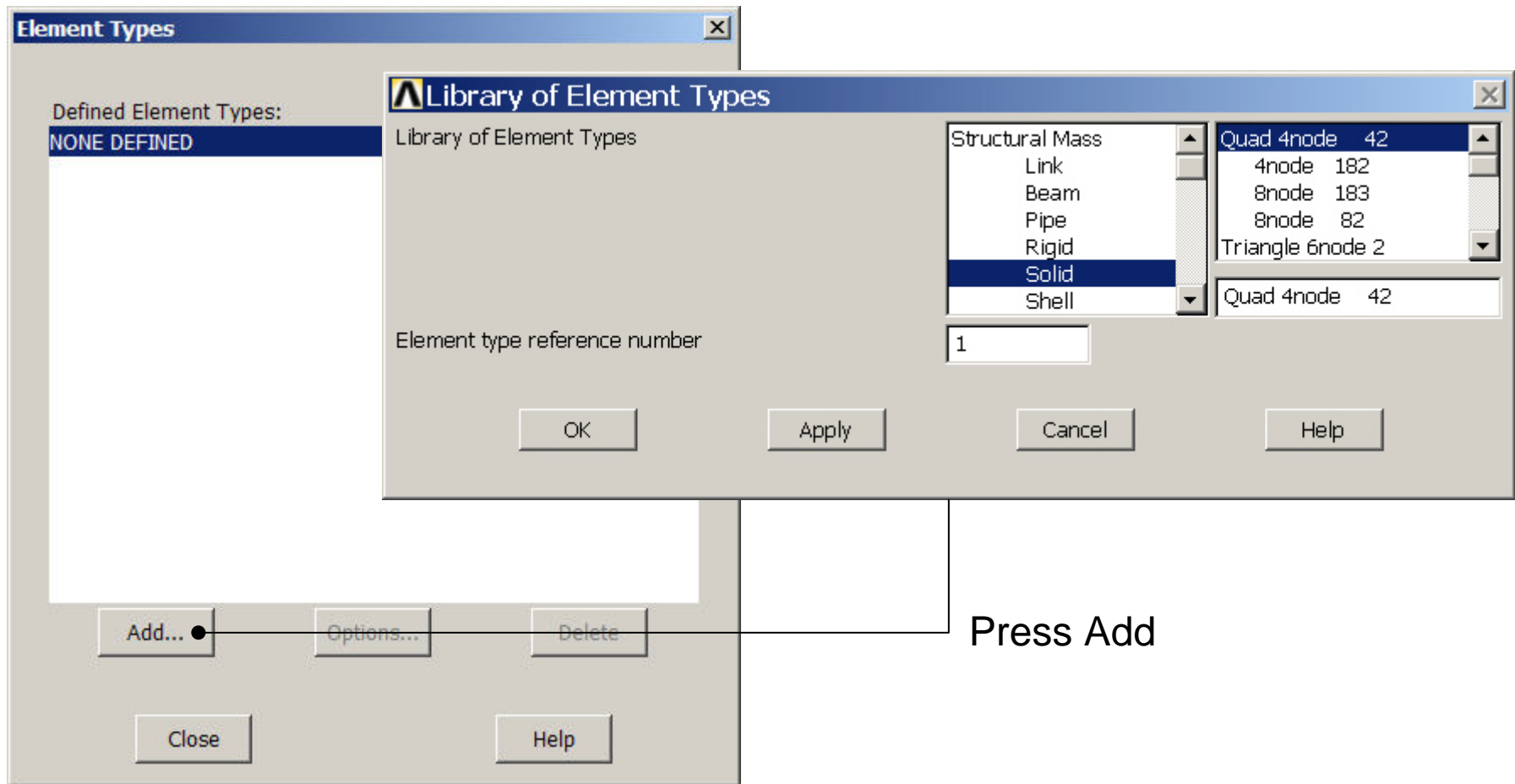






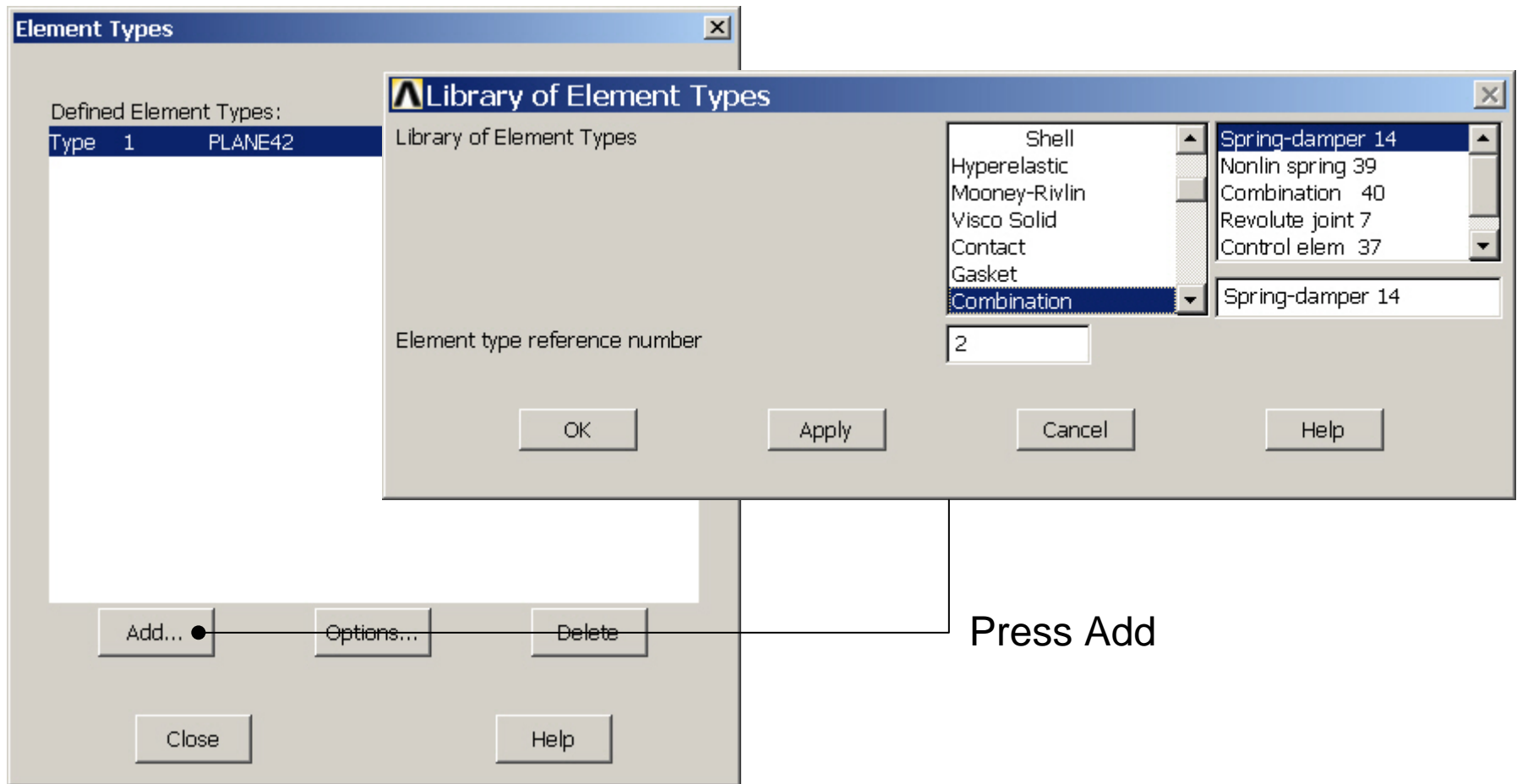
# Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



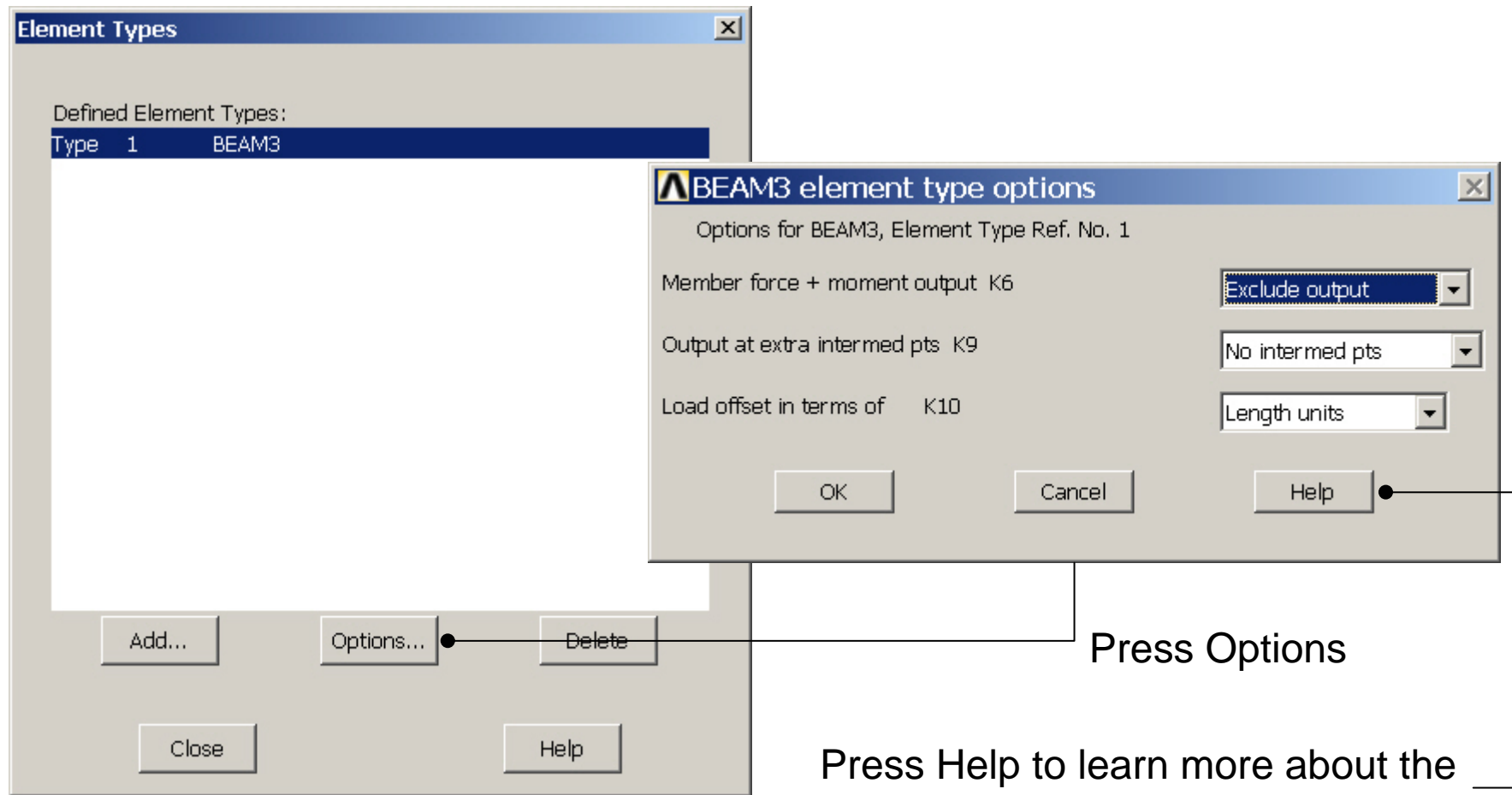
# Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



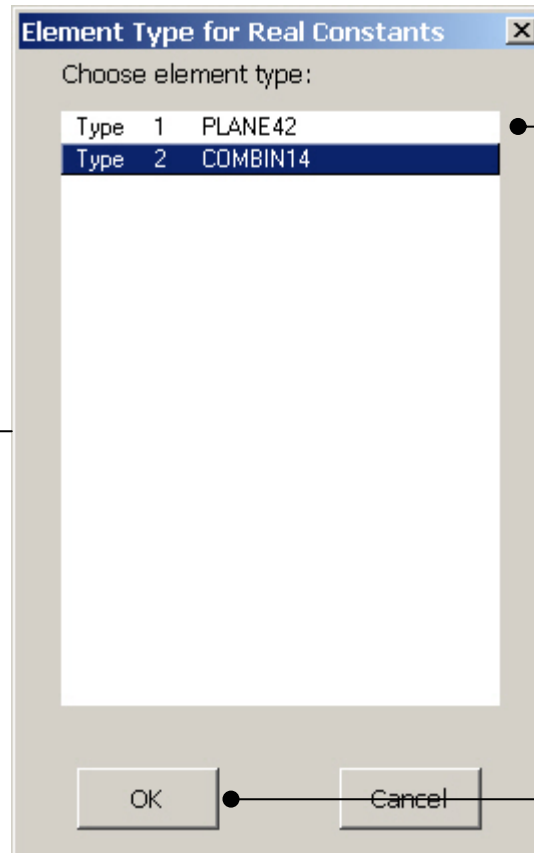
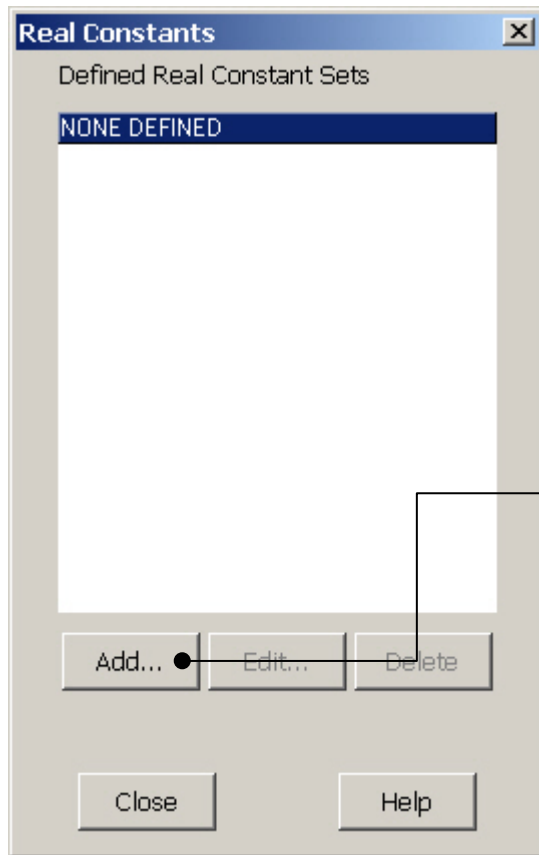
# Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



# Example – Real Constants

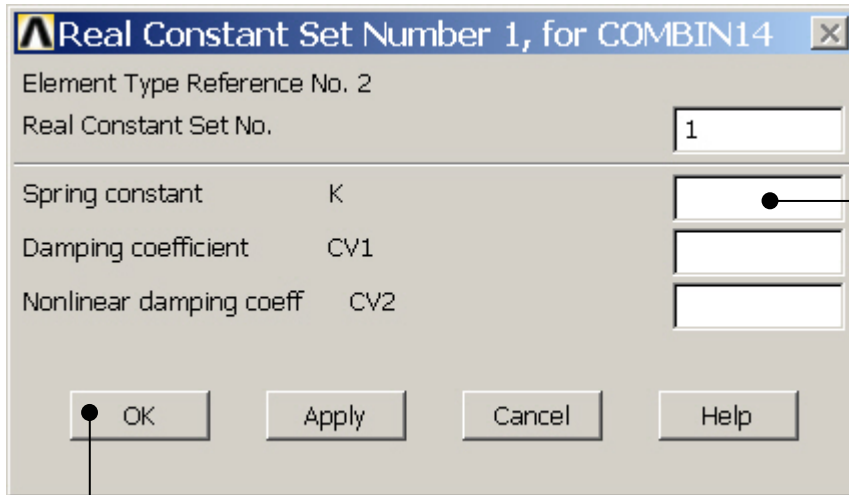
Preprocessor > Real Constants > Add



Place the cursor on the relevant element and press OK

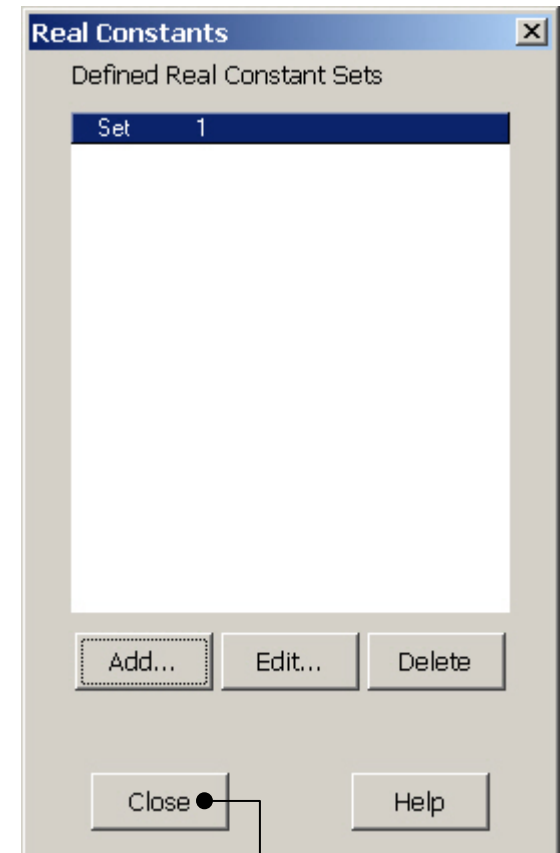
# Example - Real Constants

Preprocessor > Real Constants > Add



Enter 5 for  
Spring  
constant K

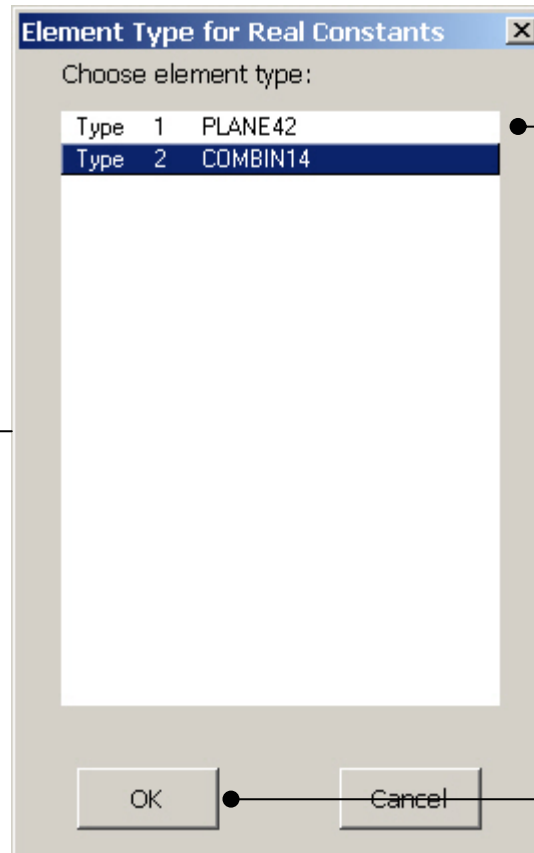
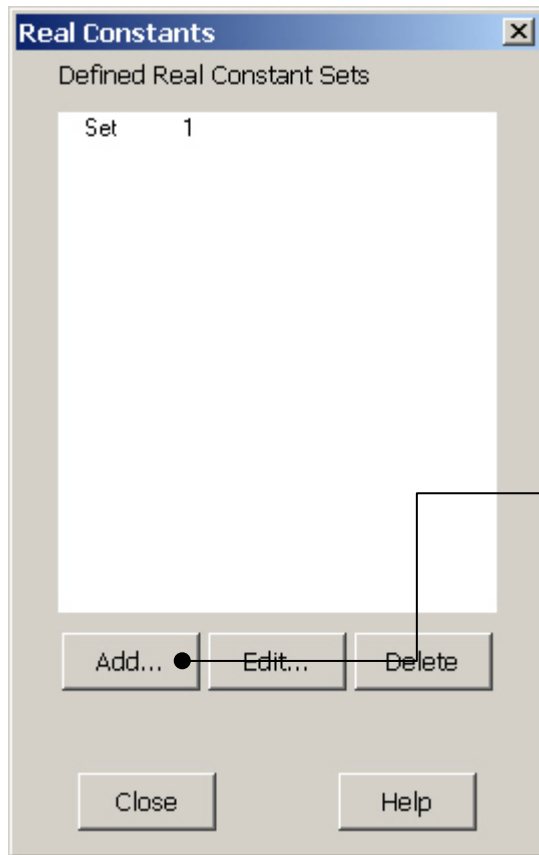
Press OK



Press Close  
to finish

# Example – Real Constants

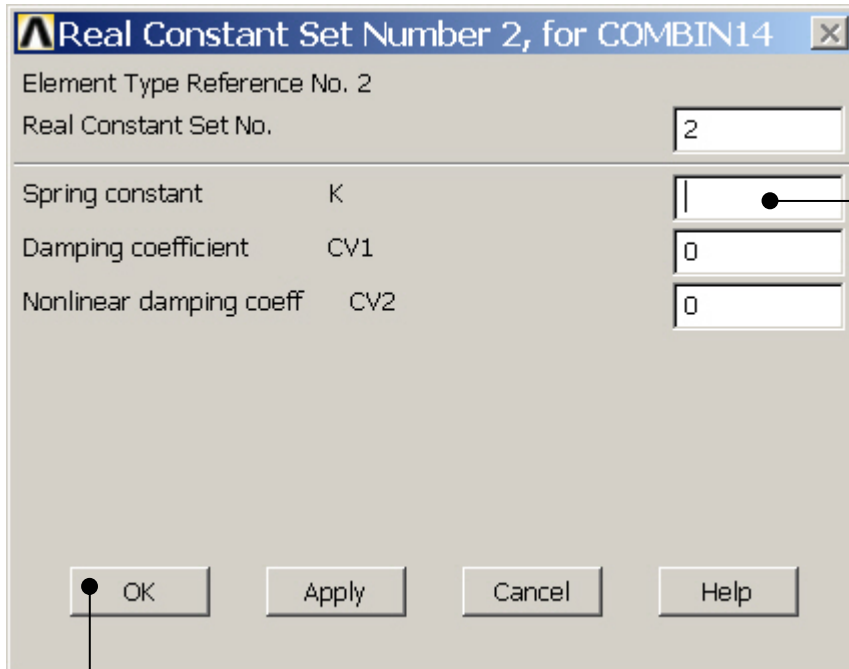
Preprocessor > Real Constants > Add



Place the cursor on the relevant element and press OK

# Example - Real Constants

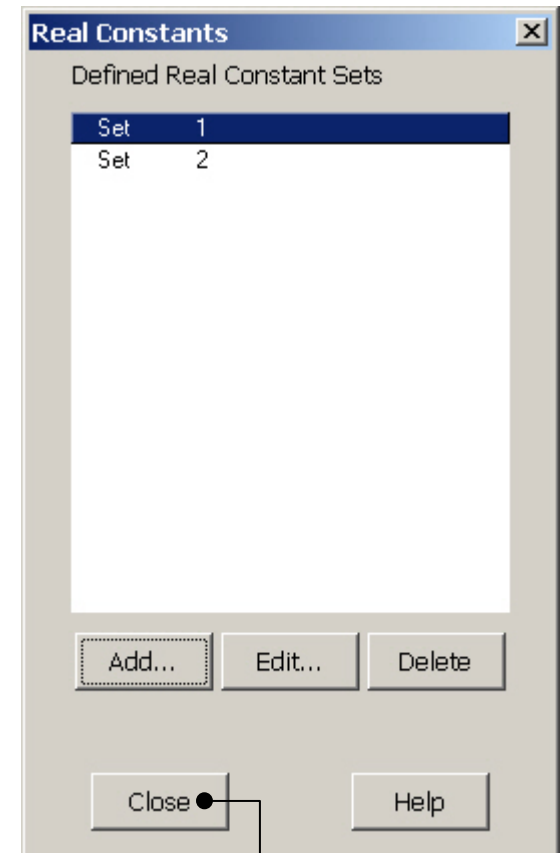
Preprocessor > Real Constants > Add



Enter 20 for  
Spring  
constant K

Press OK

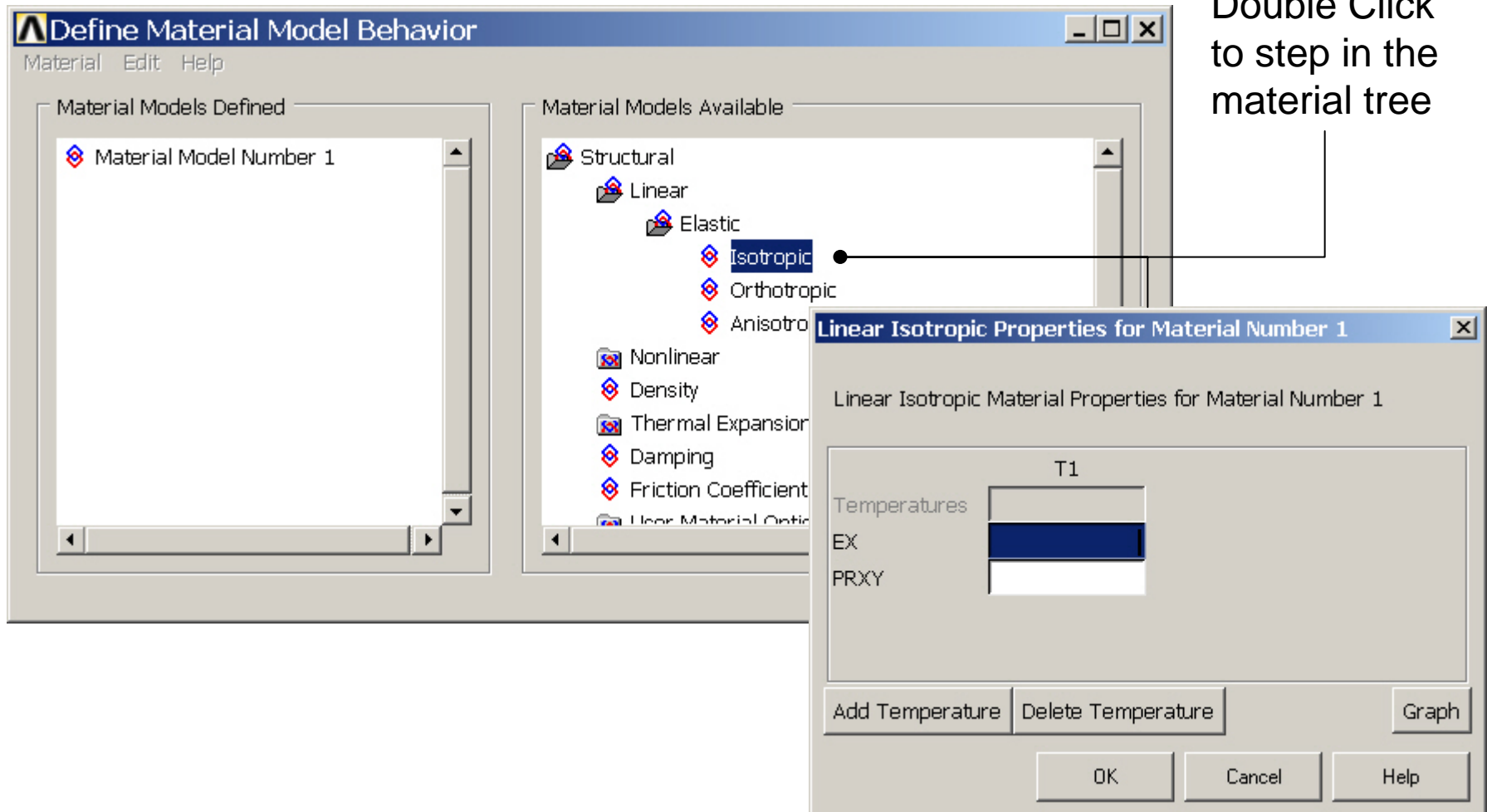
Press Close  
to finish





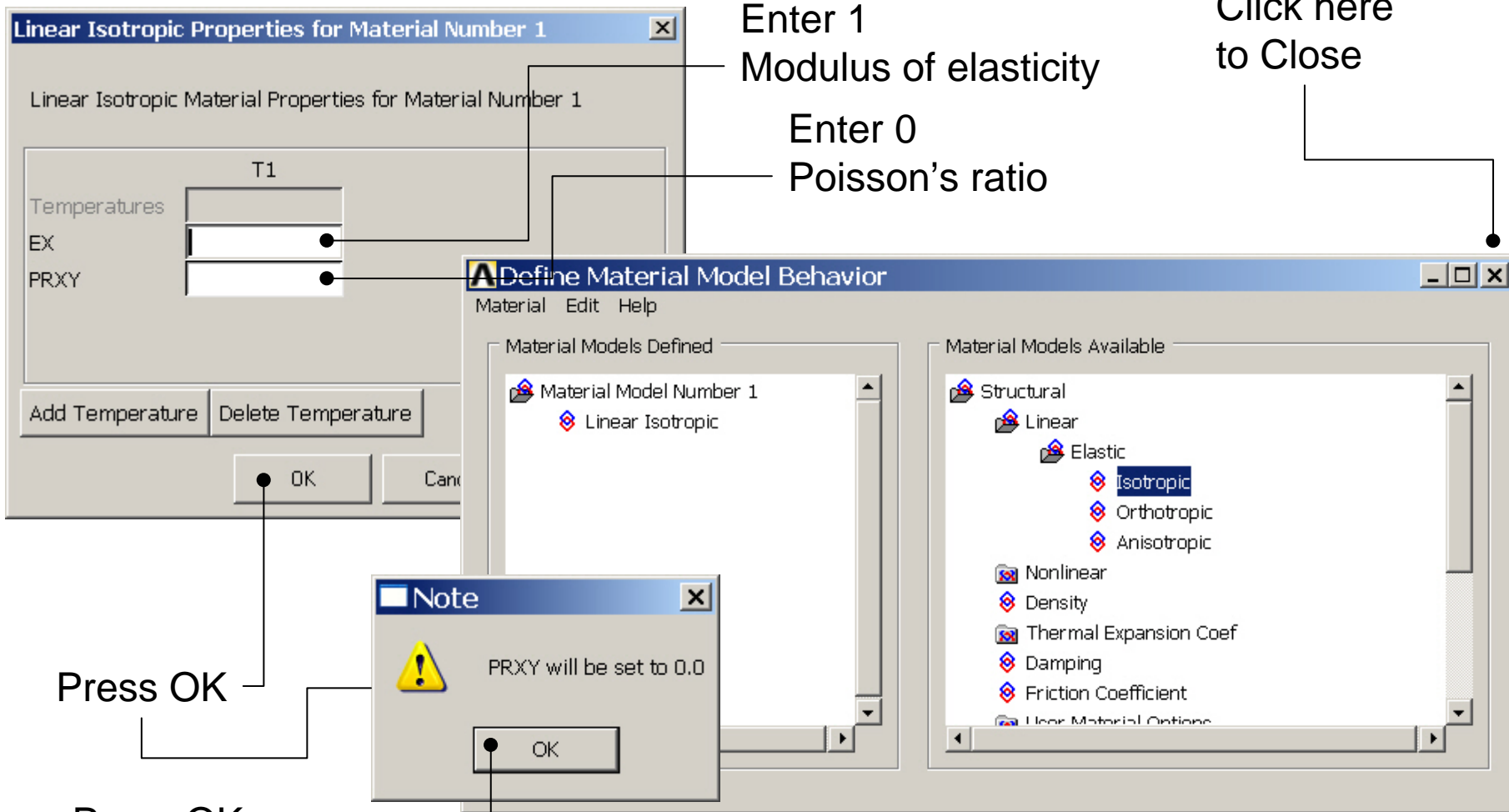
# Example - Material Properties

Preprocessor > Material Props > Material Models

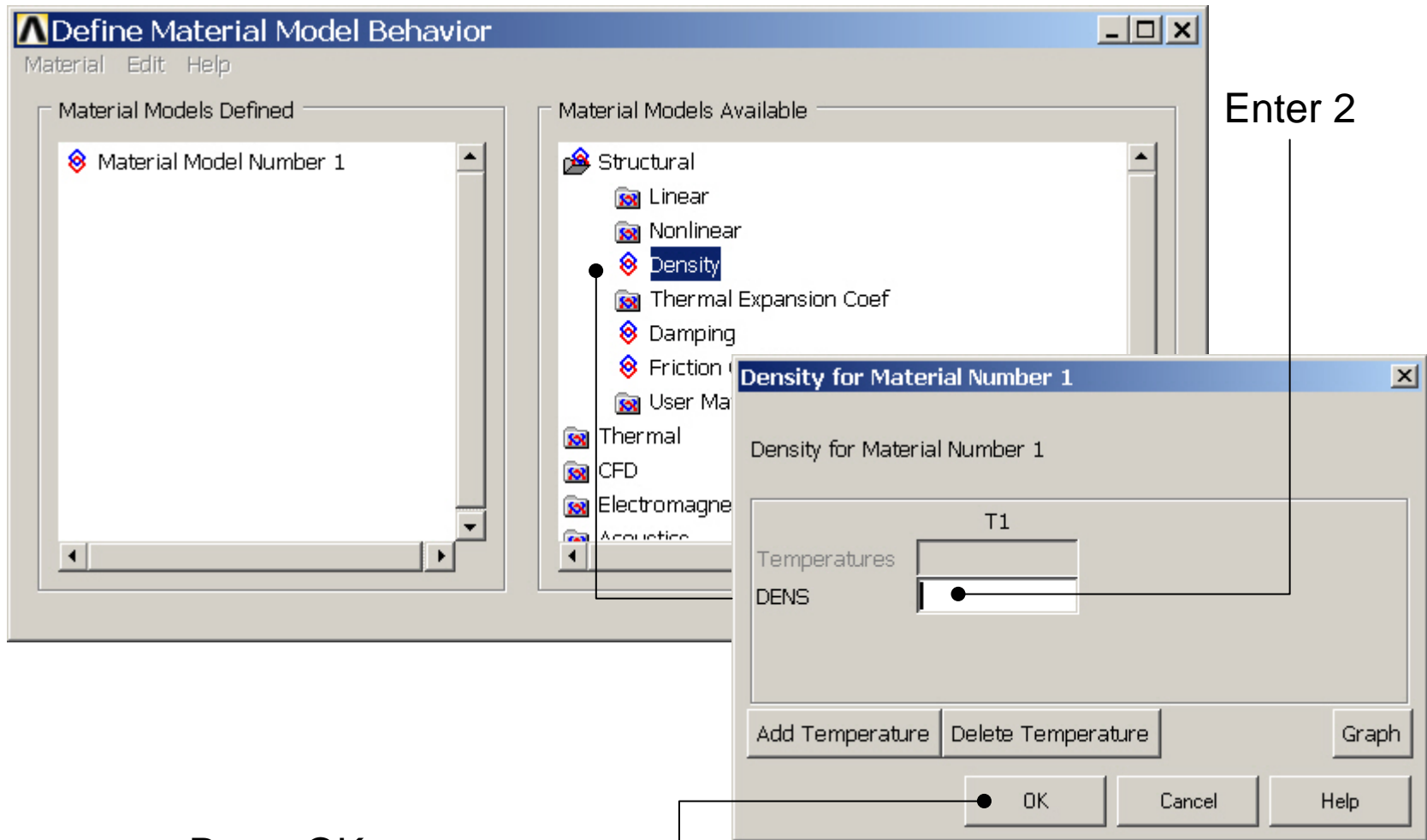


# Example - Material Properties

Preprocessor > Material Props > Material Models

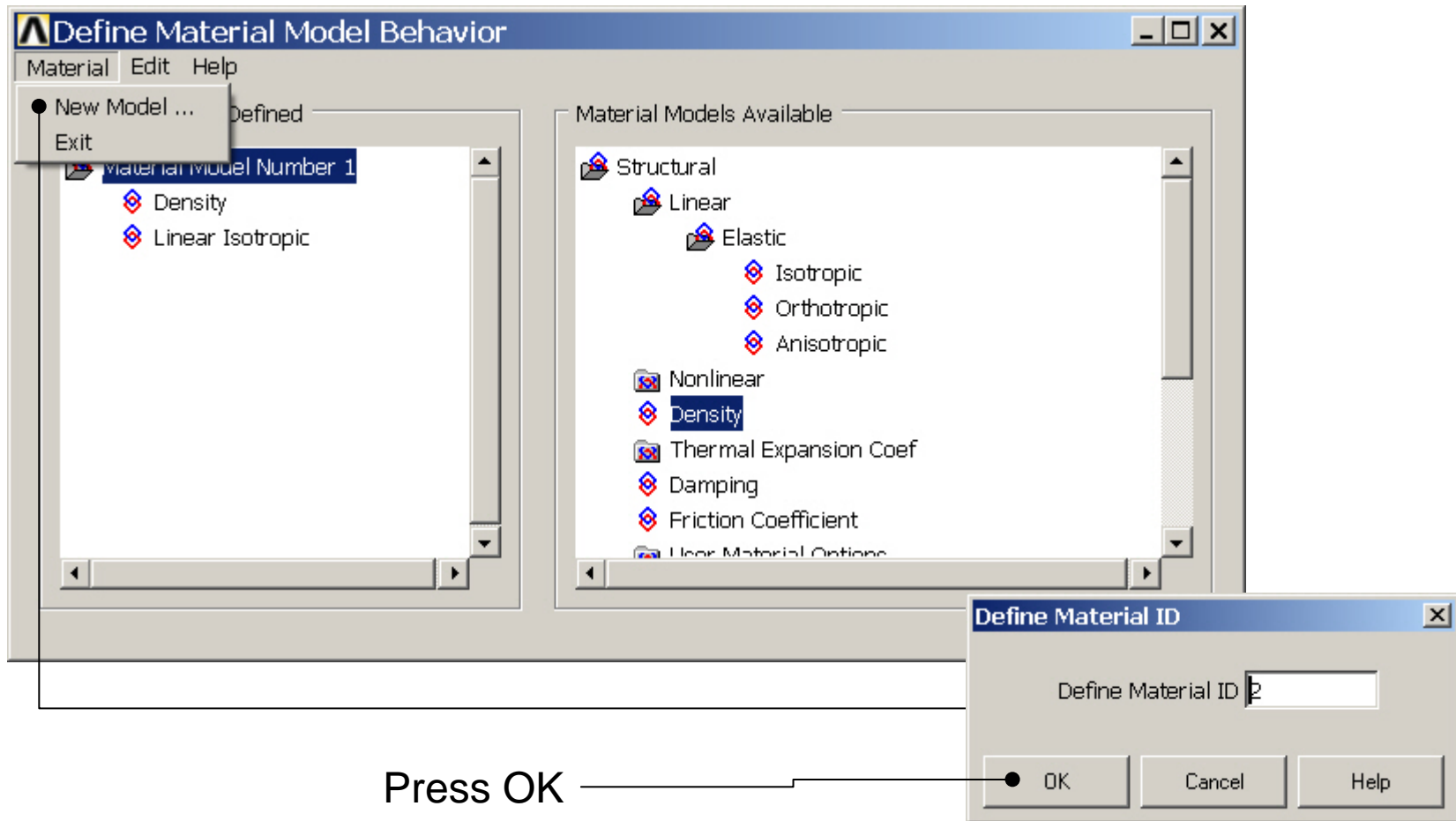


# Example - Density



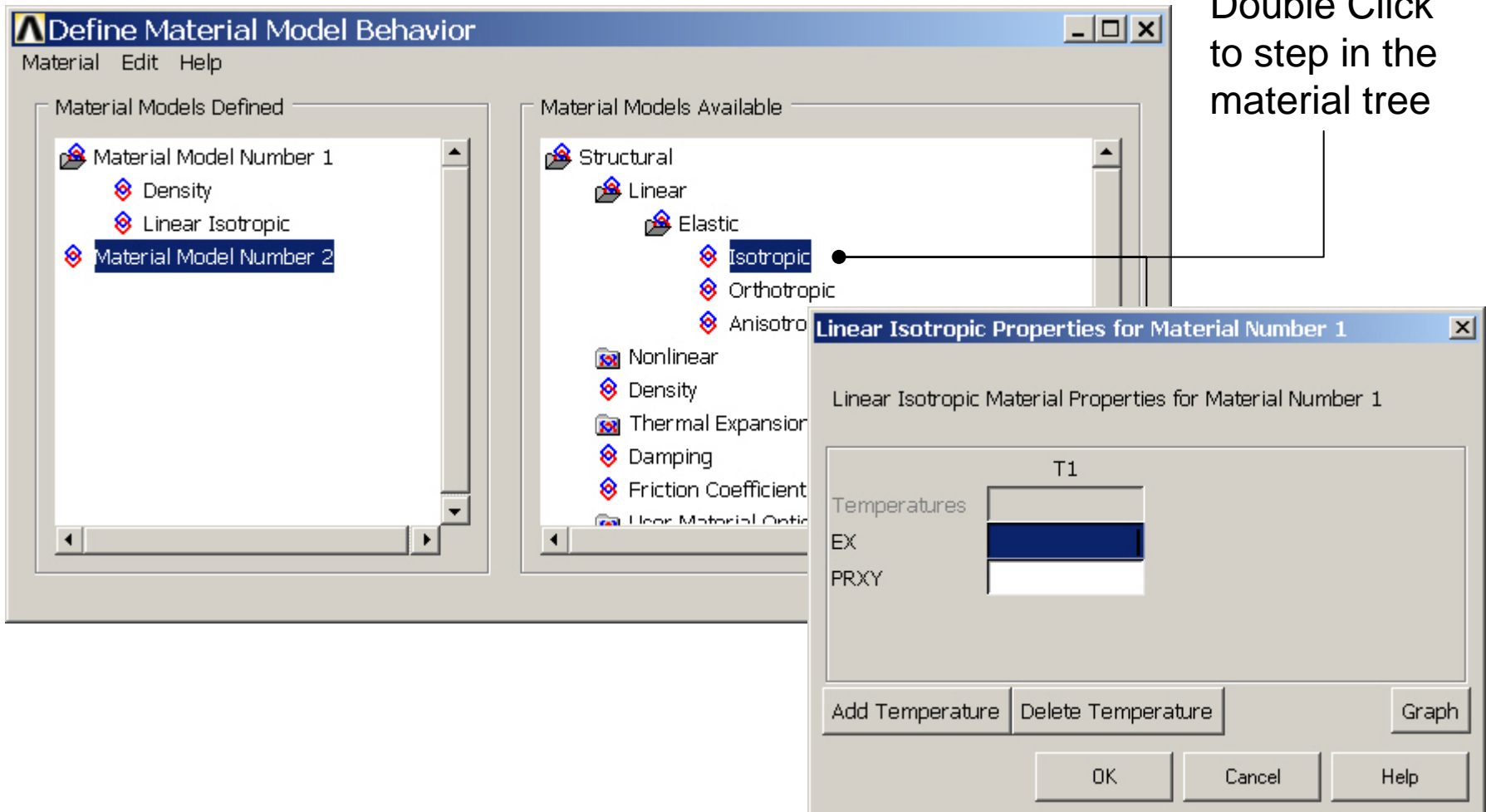
# Example - Material Properties

Preprocessor > Material Props > Material Models



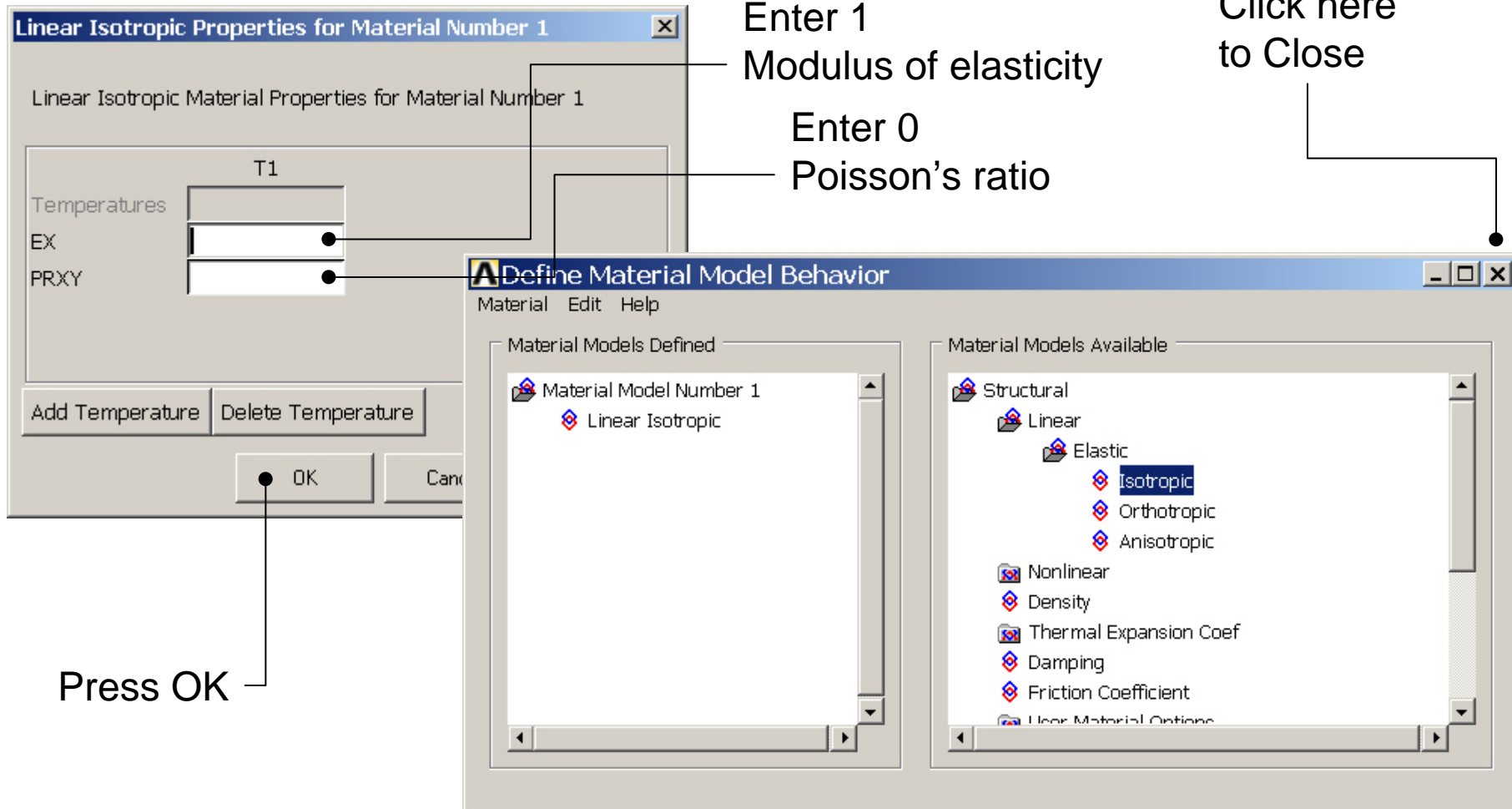
# Example - Material Properties

Preprocessor > Material Props > Material Models

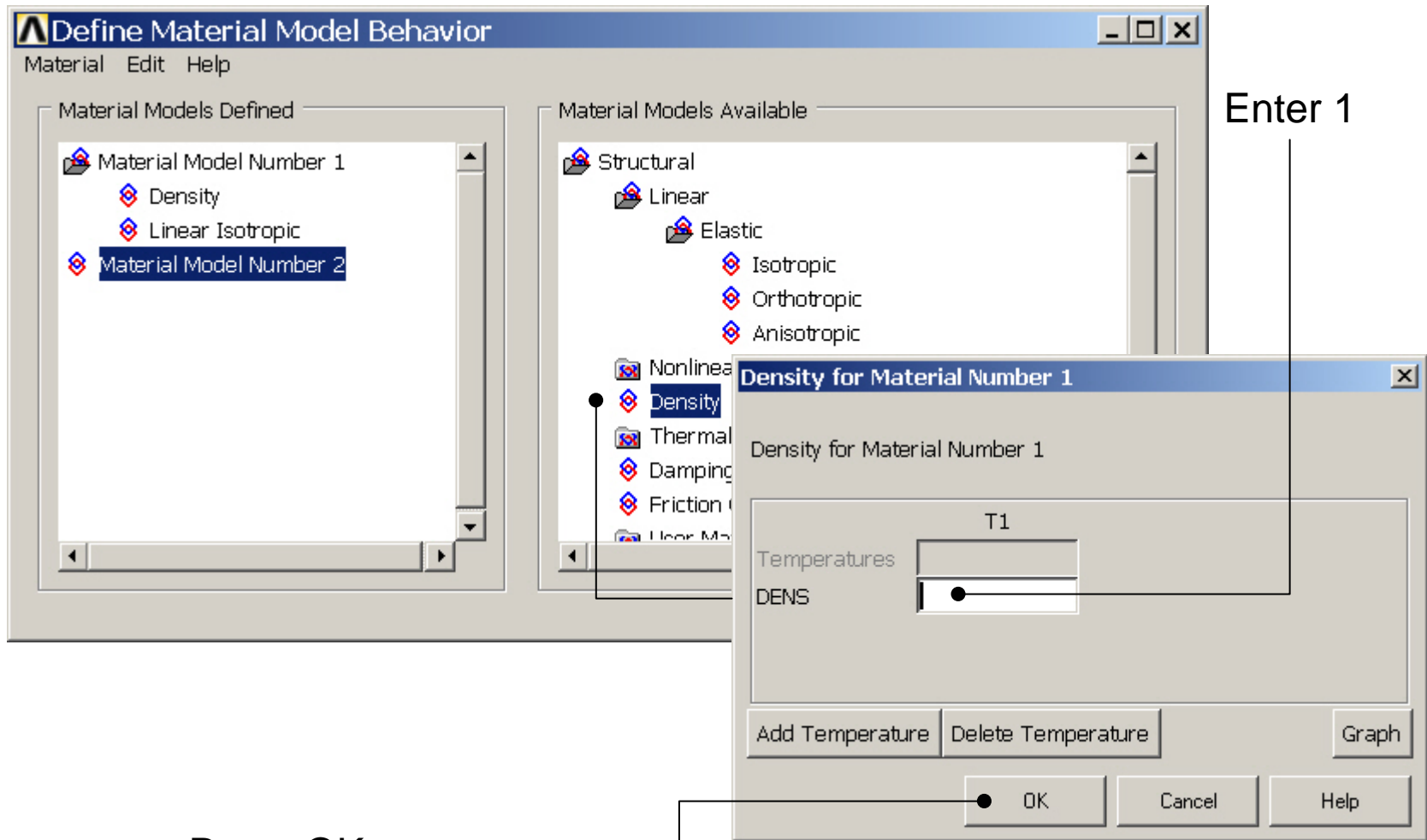


# Example - Material Properties

Preprocessor > Material Props > Material Models



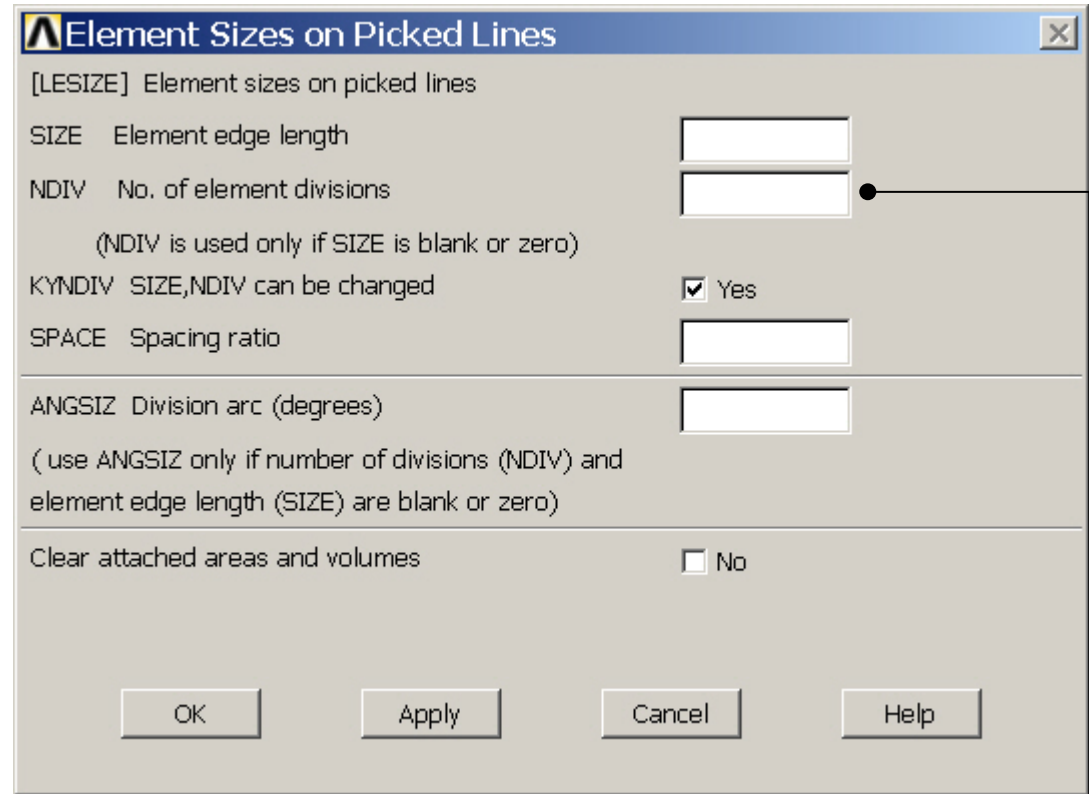
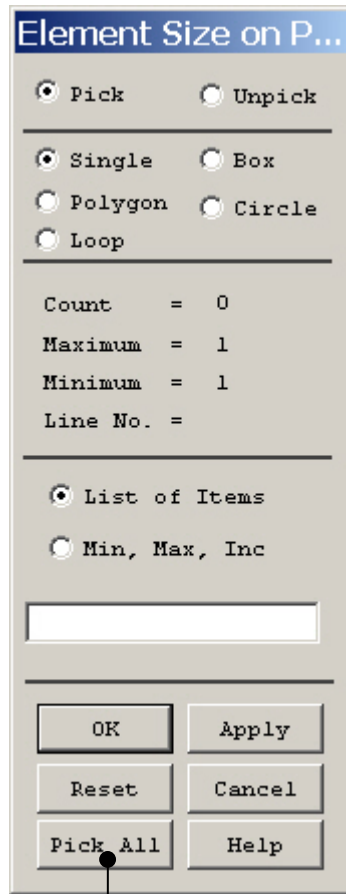
# Example - Density



# Example - Meshing

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

Select/Pick  
Lines to  
specify  
mesh size  
for



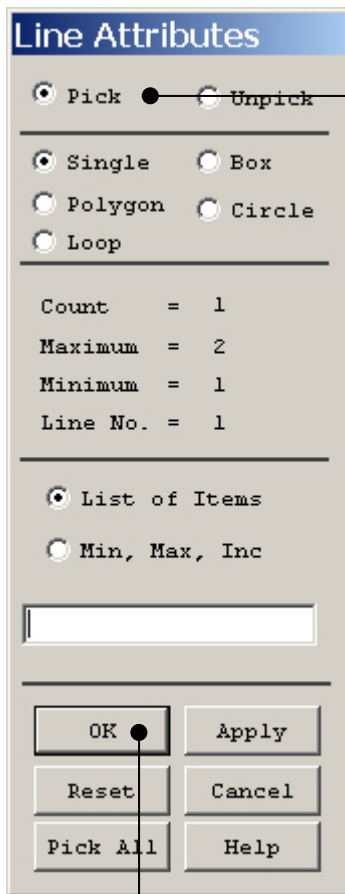
Press Pick All when finish with selection

Enter 2

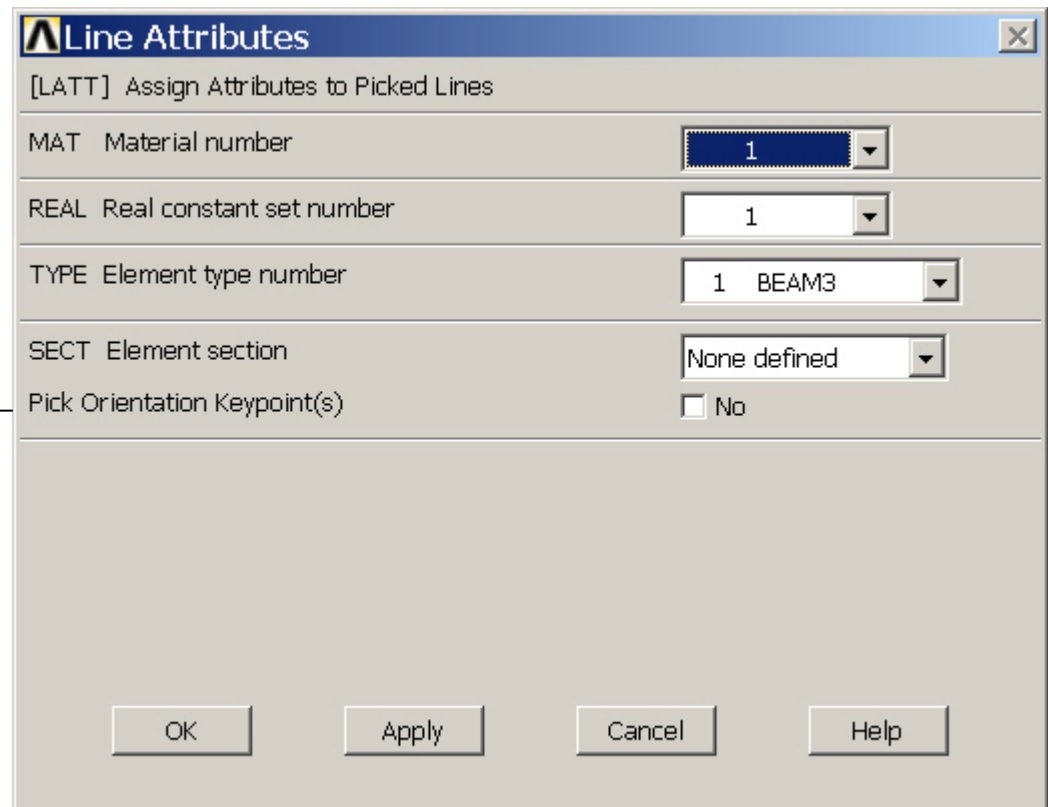


# Example – Mesh Attributes

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



Select Line 1

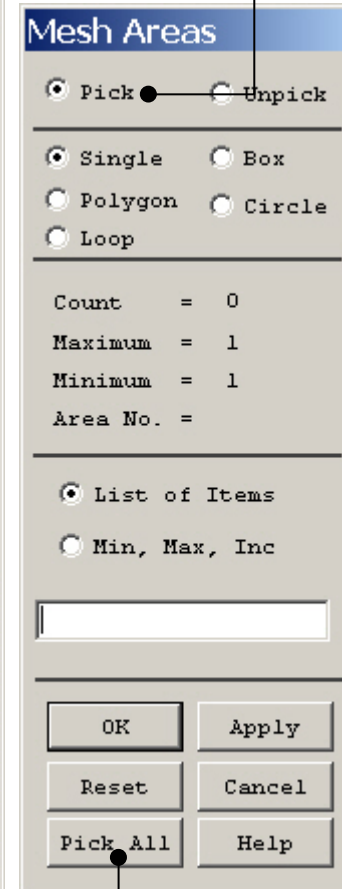


ANSYS Press OK  
Computational Mechanics, AAU, Esbjerg

Example0430

# 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 – Create Node

Create Node on KP

☒ Pick      ☐ Unpick

---

☒ Single      ☐ Box  
☐ Polygon      ☐ Circle  
☐ Loop

---

Count      =      0  
Maximum    =      9  
Minimum    =      1  
KeyP No.   =

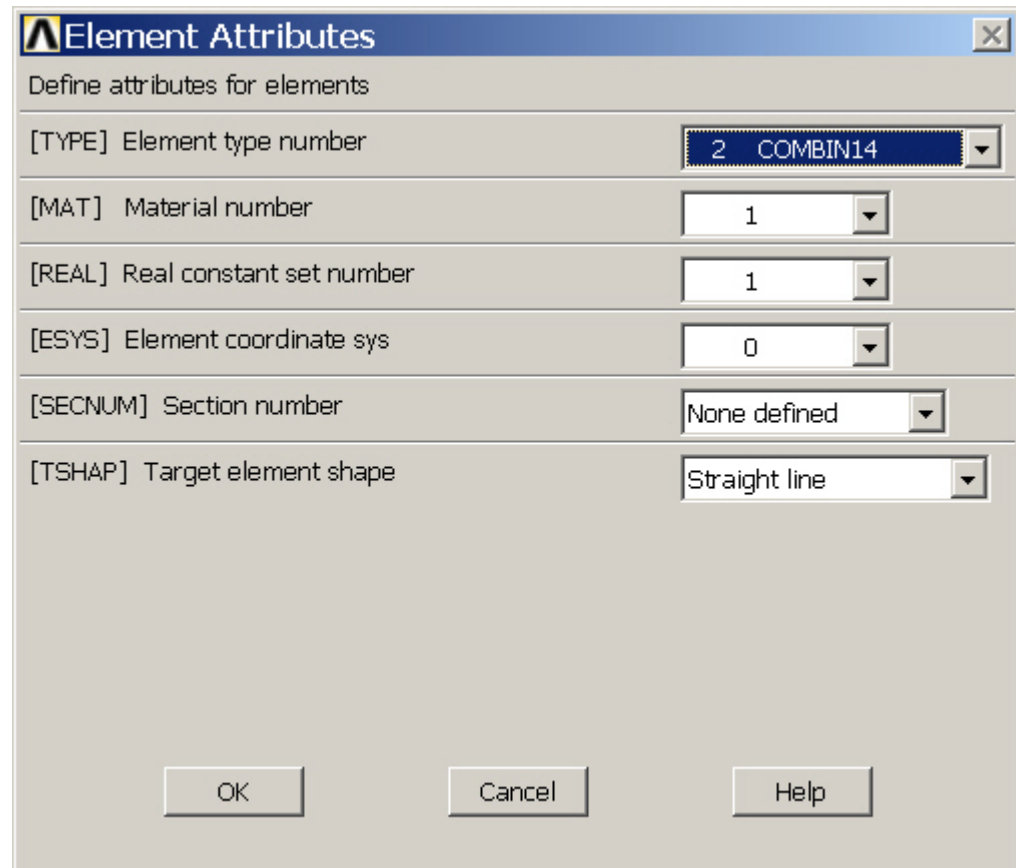
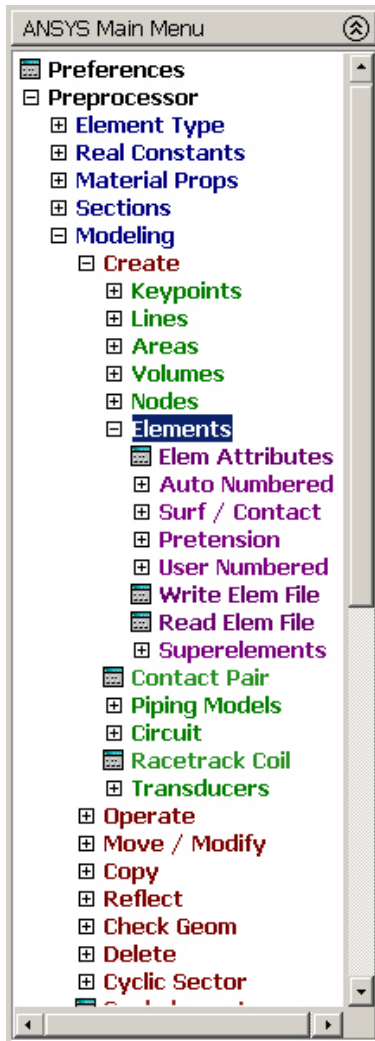
---

☒ List of Items  
☐ Min, Max, Inc

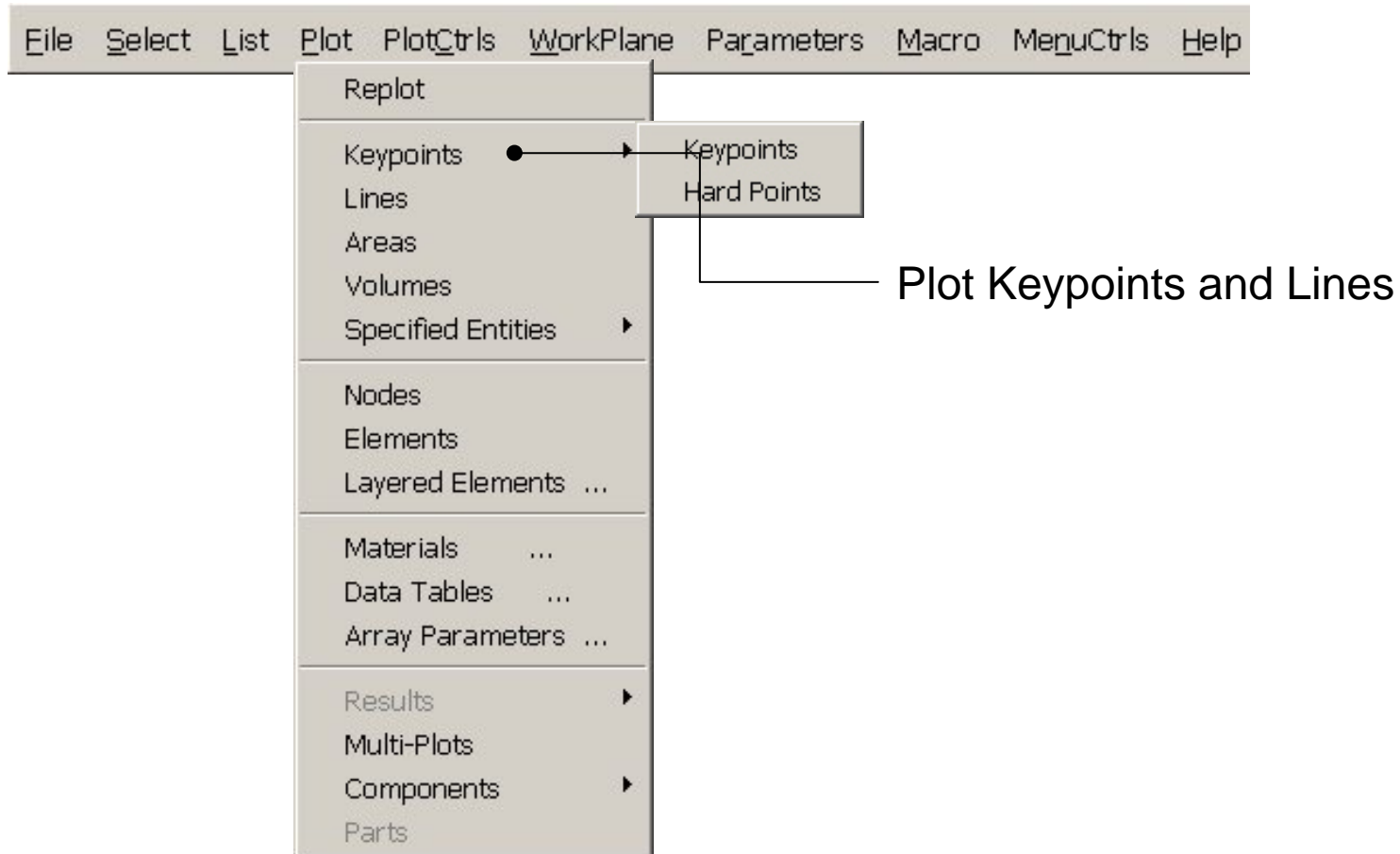
---

OK      Apply  
Reset      Cancel  
Pick All      Help

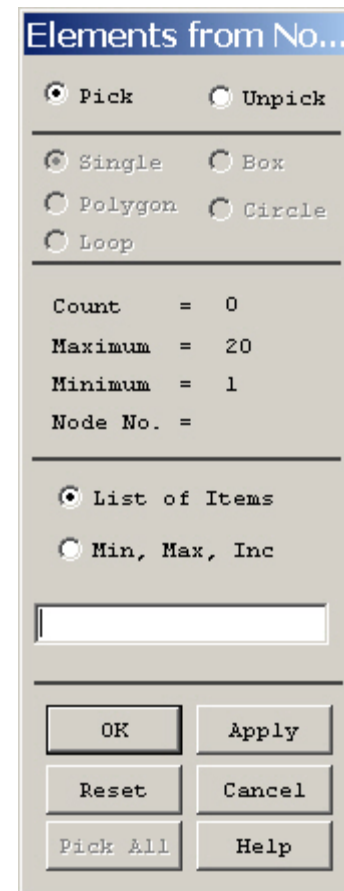
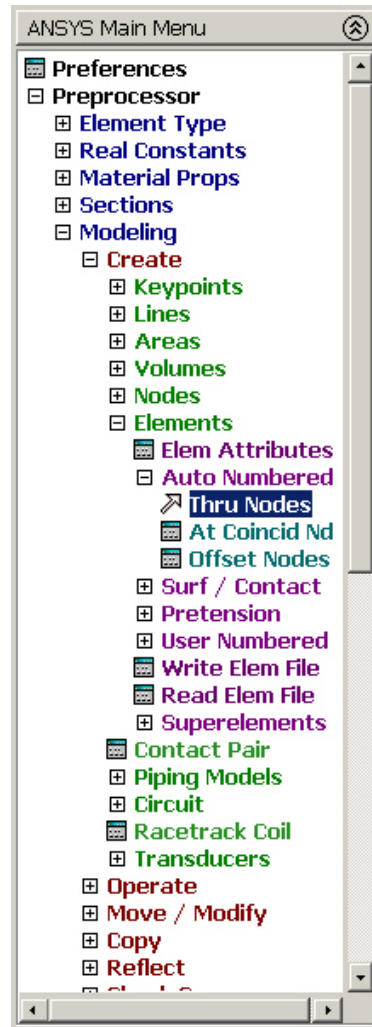
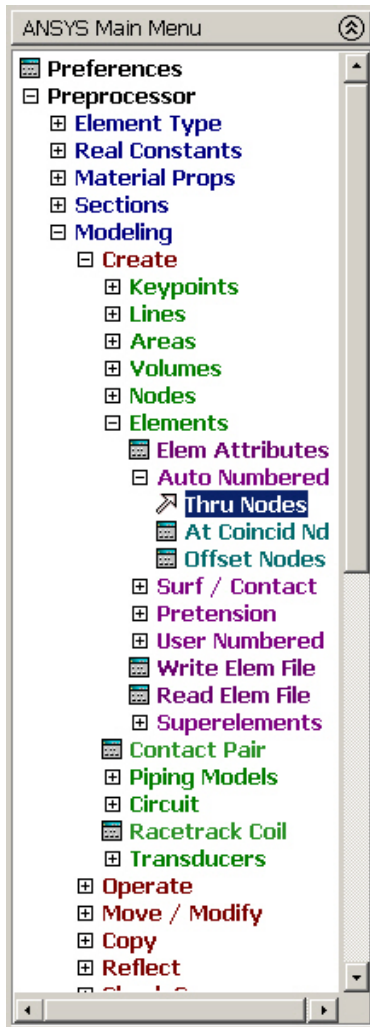
# Example – Element attributes



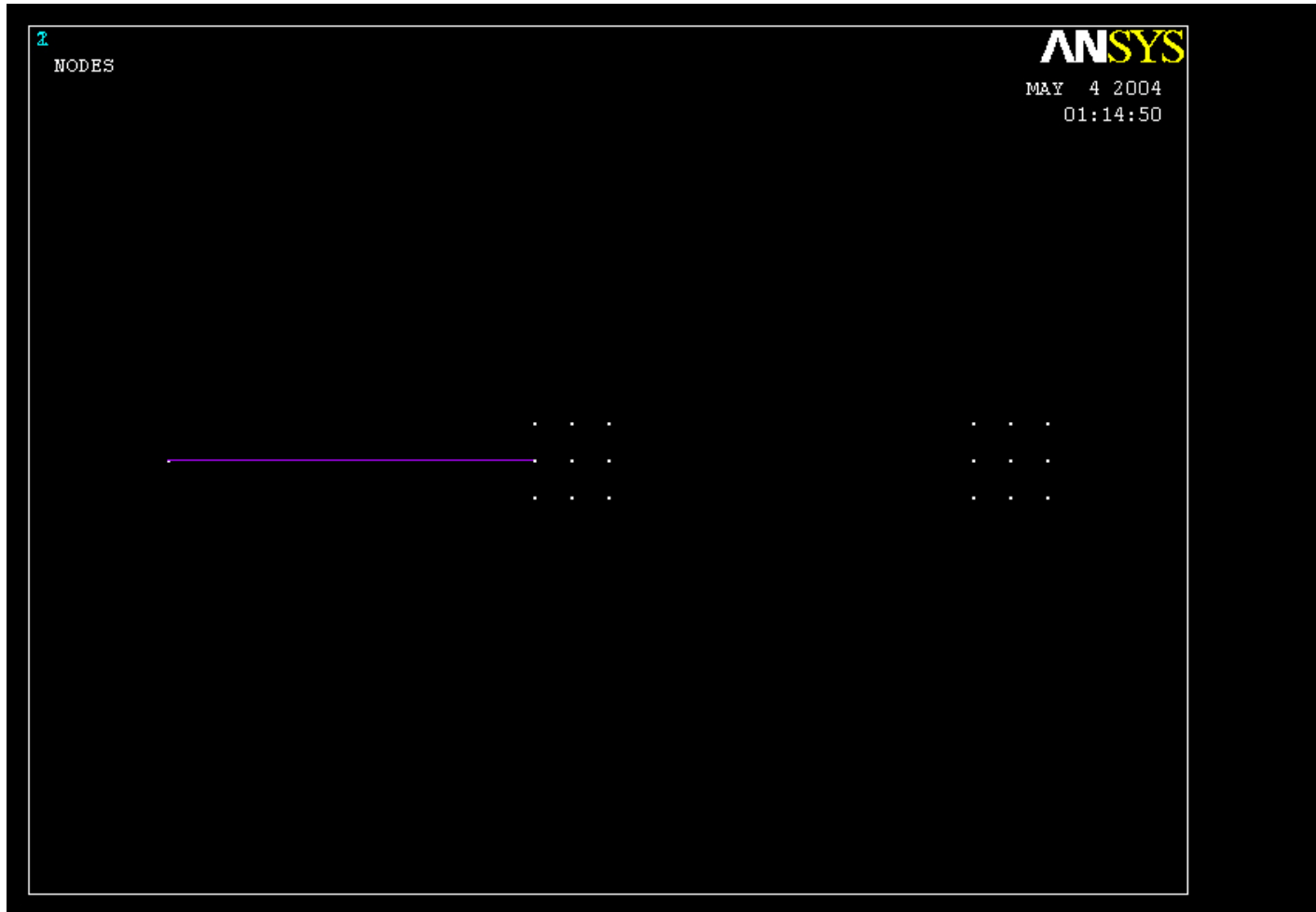
# Example - Plot



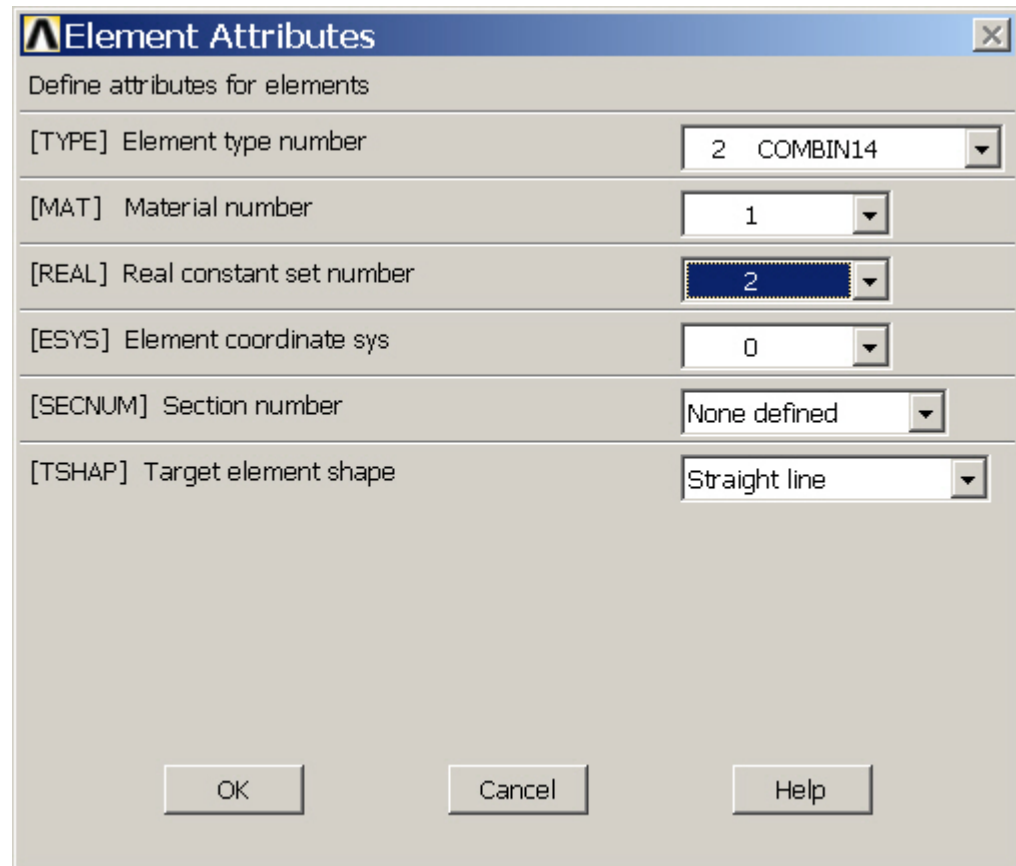
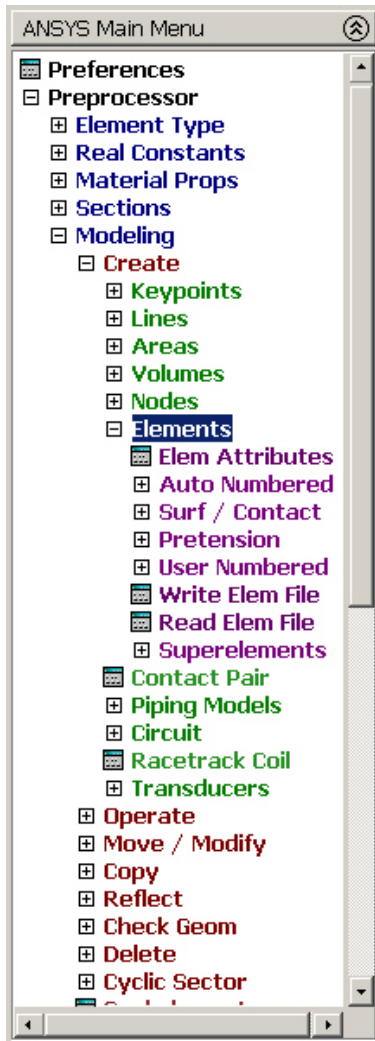
# Example – Create Elements



# Example – Create Elements

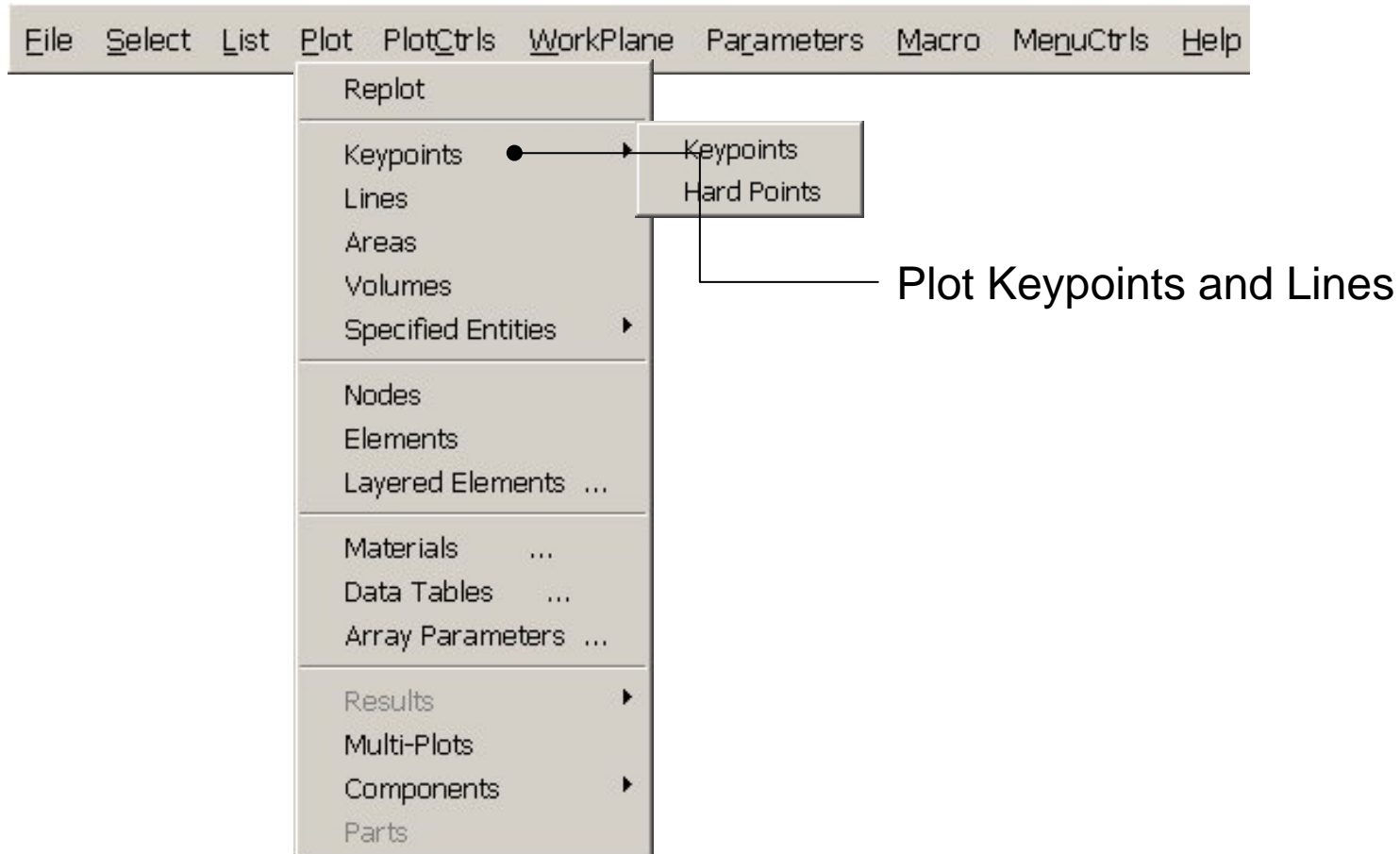


# Example – Element attributes

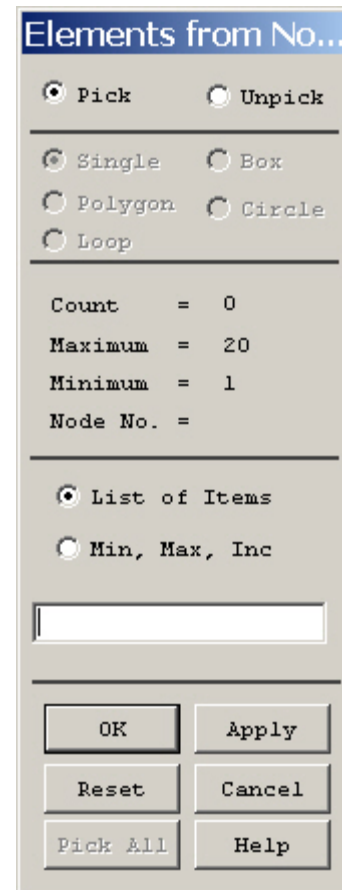
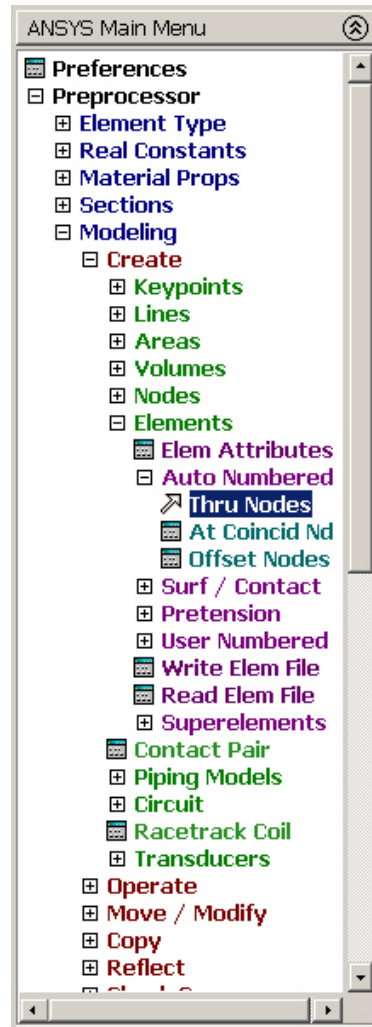
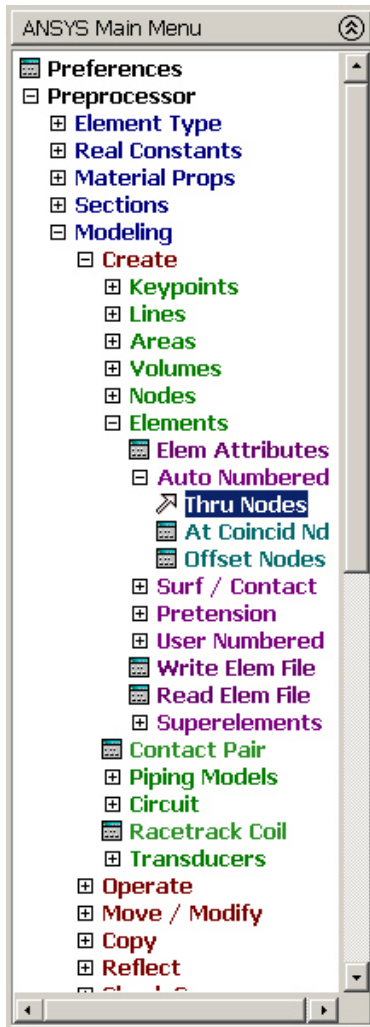




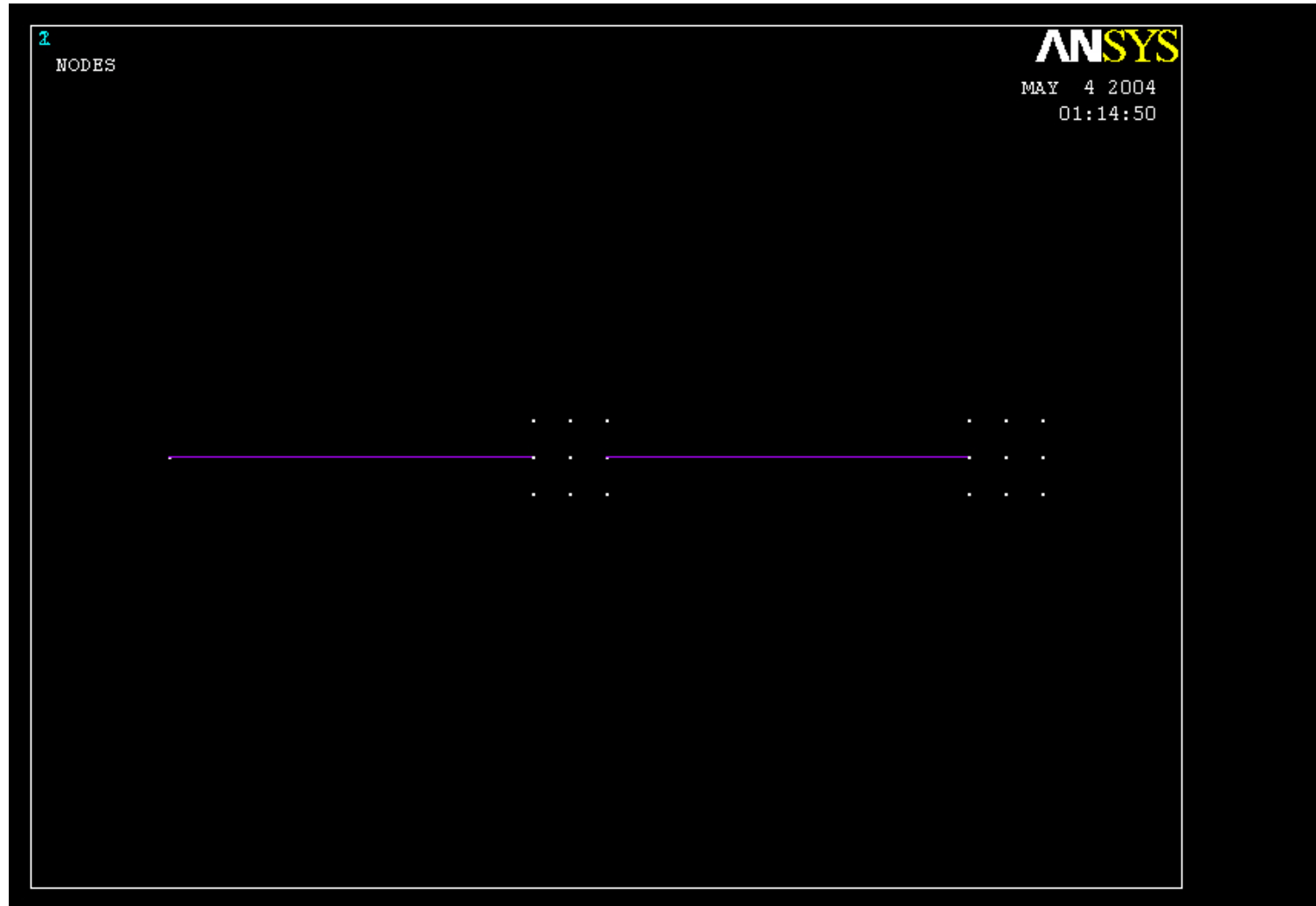
# Example - Plot



# Example – Create Elements

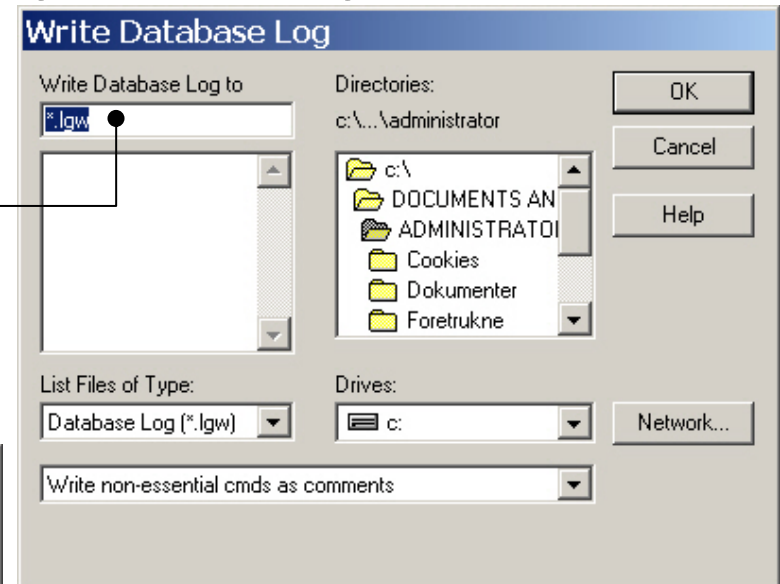


# Example – Create Elements

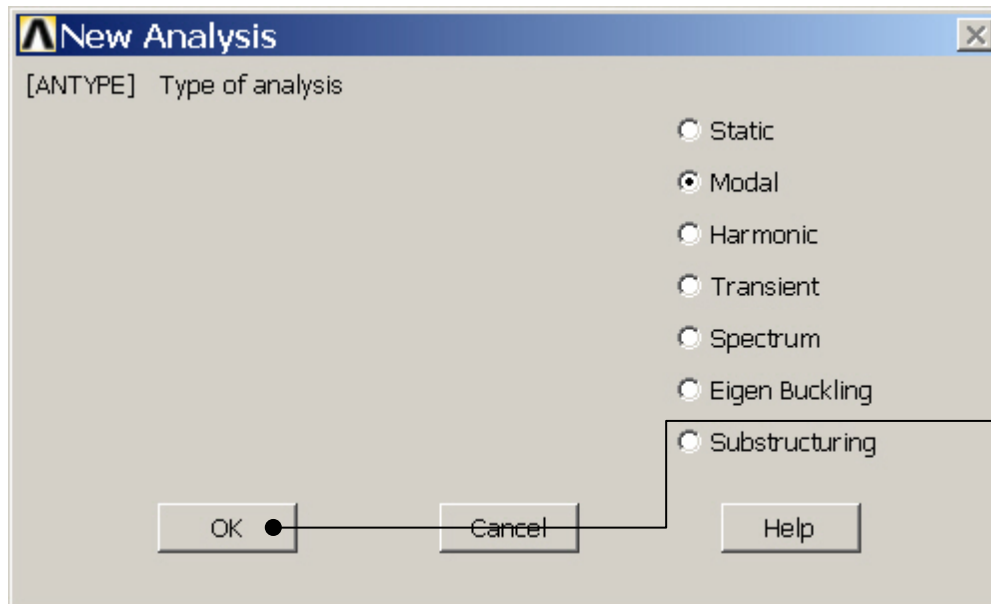


# Example – Analysis Type

**File > Write DB log file**  
Enter “example0430.lgw”



**Solution > Analysis Type > New Analysis**



Press OK

# Example – MA Analysis Options

The image shows the ANSYS Main Menu on the left and the Modal Analysis dialog box on the right. The dialog box is titled "Modal Analysis" and contains several sections for configuring the analysis options. Annotations with arrows point to specific elements in the dialog box:

- Select Subspace:** Points to the "Block Lanczos" radio button in the "[MODEPT] Mode extraction method" section.
- Enter 5:** Points to the "No. of modes to extract" input field, which currently contains the value "0".
- Enter 5:** Points to the "Expand mode shapes" checkbox, which is checked "Yes".
- Press OK:** Points to the "OK" button at the bottom of the dialog box.

The ANSYS Main Menu on the left includes the following options:

- Preferences
- Preprocessor
- Solution
  - Analysis Type
    - New Analysis
    - ExpansionPass
    - Analysis Options
  - Define Loads
  - Load Step Opts
  - Solve
  - FSI Set Up
  - Unabridged Menu
- General Postproc
- TimeHist Postpro
- Topological Opt
- RDM Tool
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

# Example – Subspace Options

**Subspace Modal Analysis**

[MODOPT] Mode Extraction Options

FREQB Start Freq (initial shift)

FREQE End Frequency

Nrmkey Normalize mode shapes

[RIGID] Known rigid body modes

All DOF  
UX  
UY  
ROTZ

[SUBOPT] Subspace iteration options

SUBSIZ Subspace working size

NPAD No. of extra vectors

NPERBK No of modes/memory block

Number of subspace iterations

NUMSSI Maximum number

NSHIFT Min, before shift

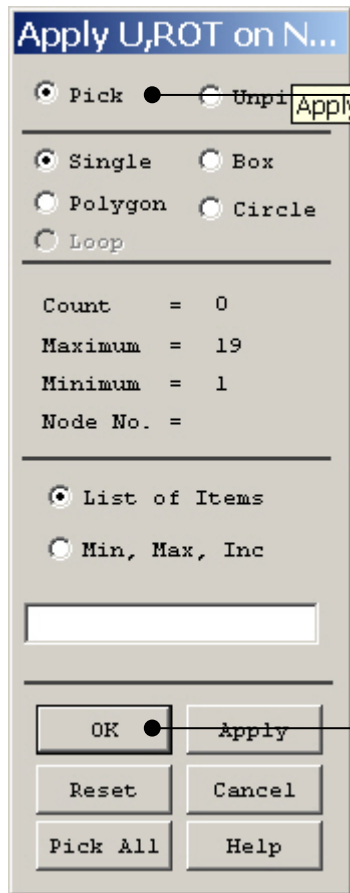
Strmck Sturm sequence check

OK Cancel Help

Press OK

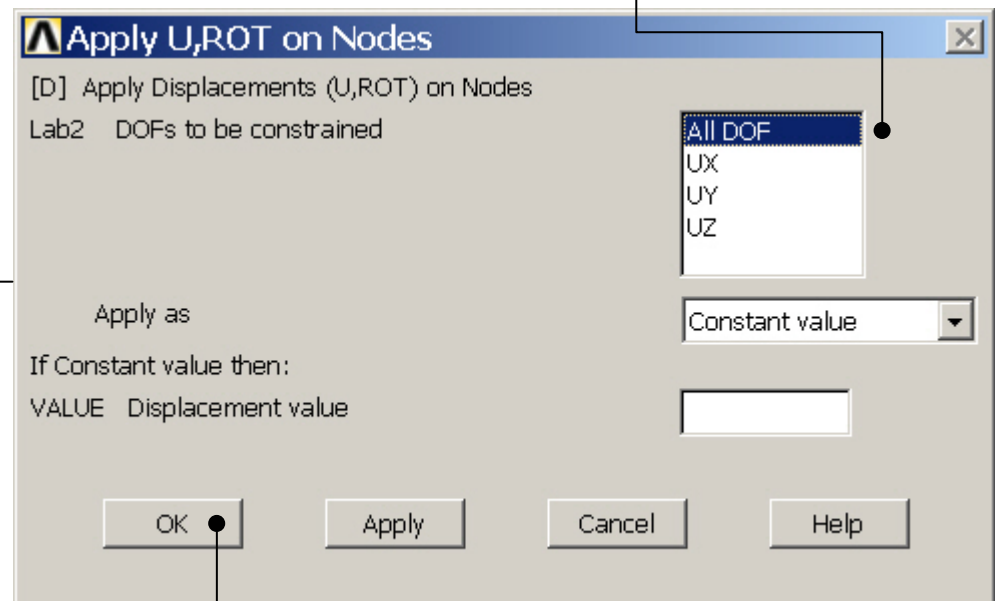
# Example – Define Loads

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



Select the left single node

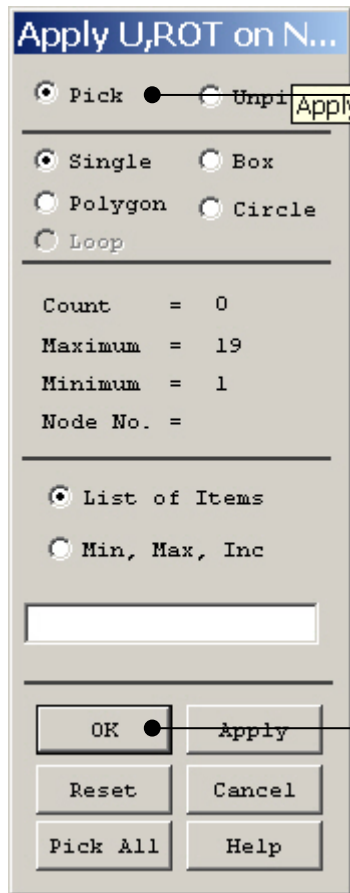
Select All DOF to fix/clamp the system



Press OK

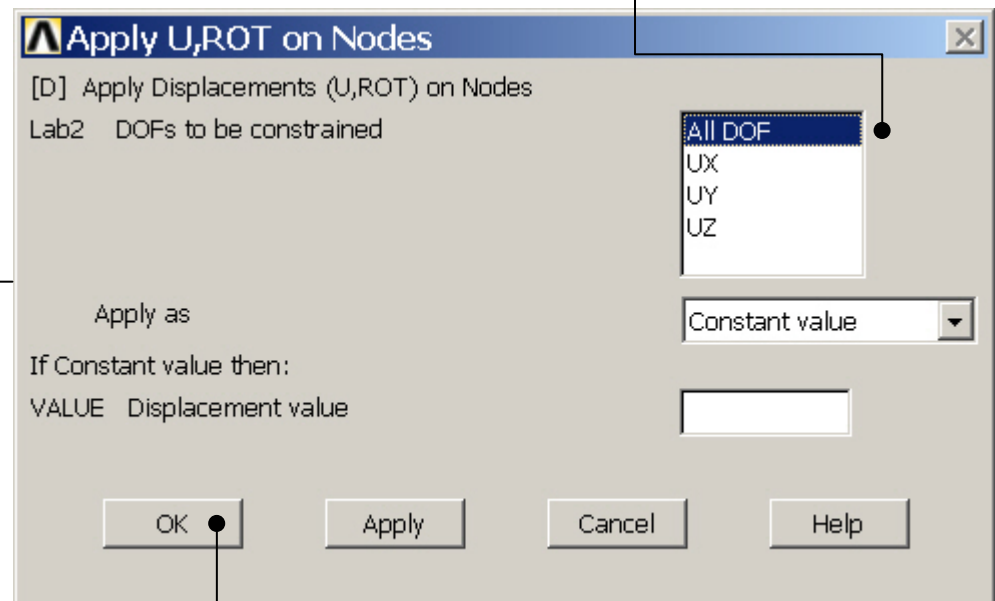
# Example – Define Loads

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



Select the left single node

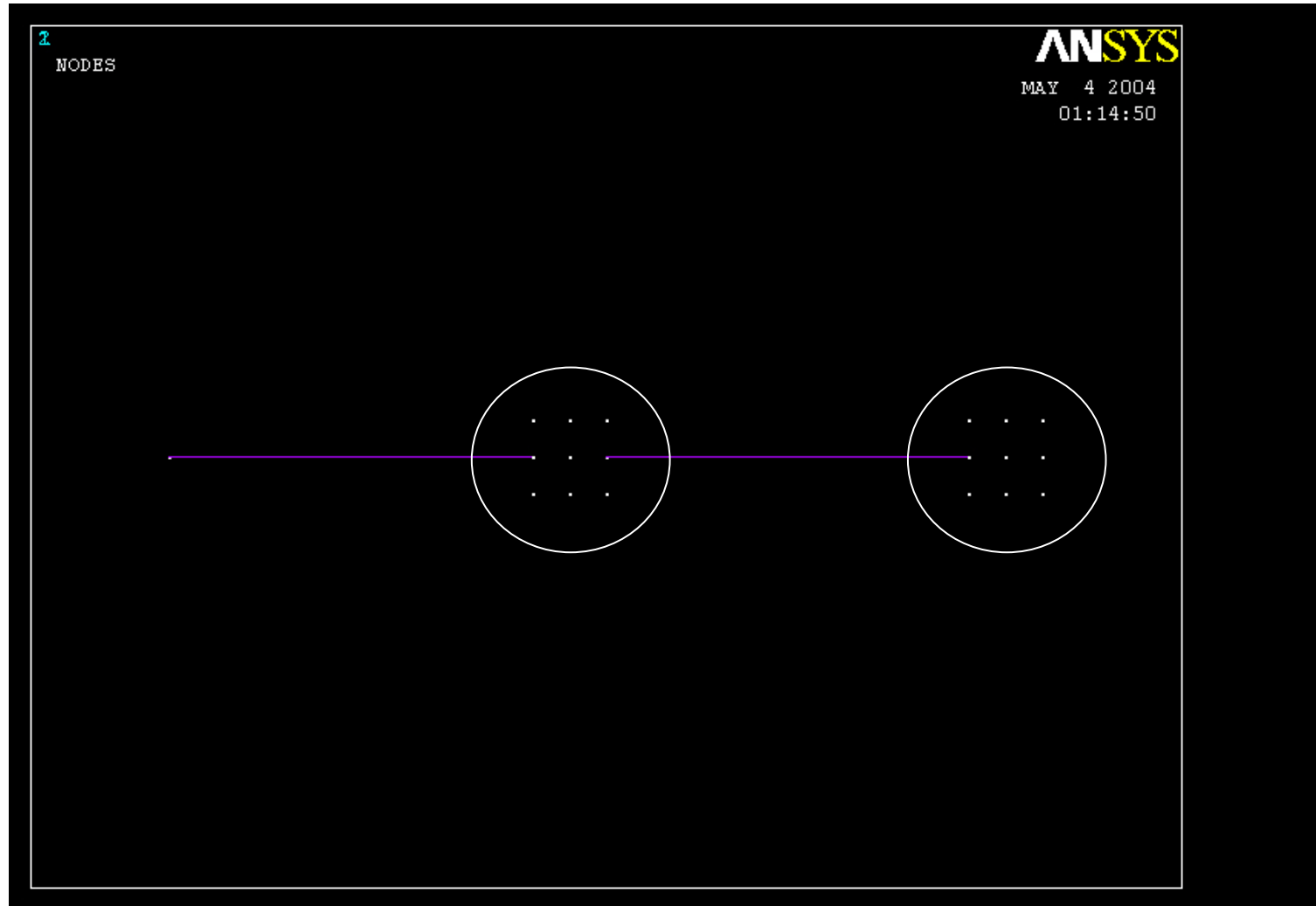
Select UY and UZ to move the system  
In the x-direction only



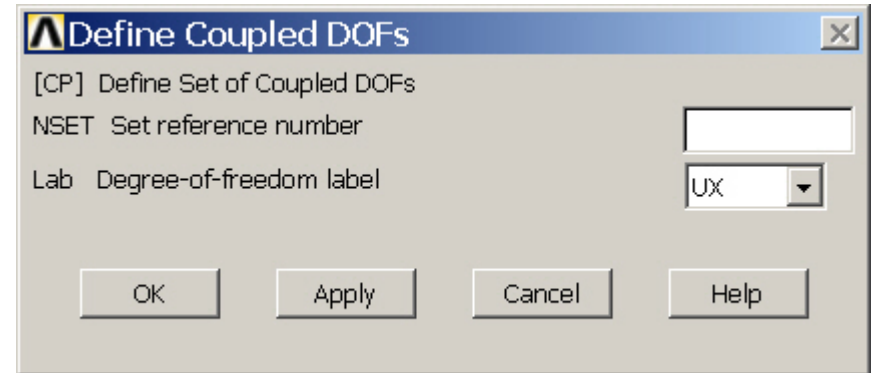
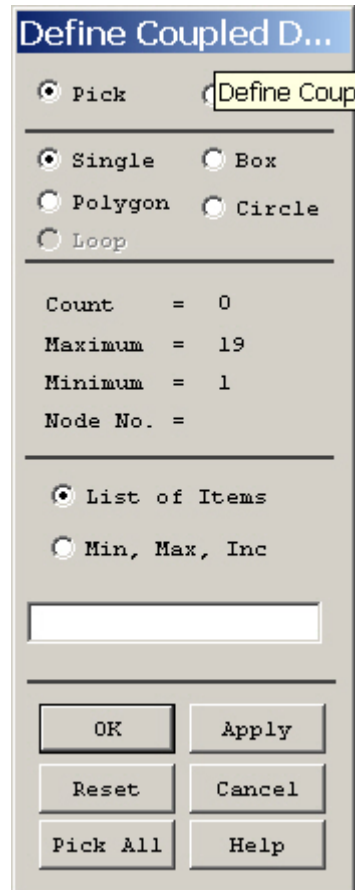
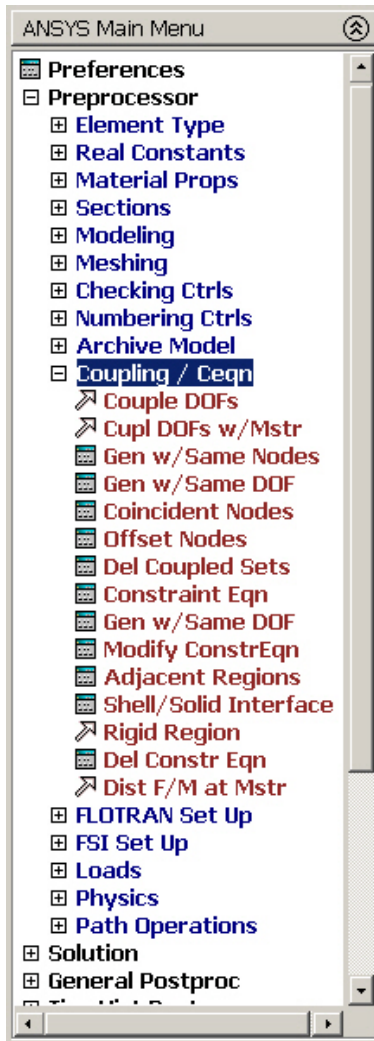
Press OK



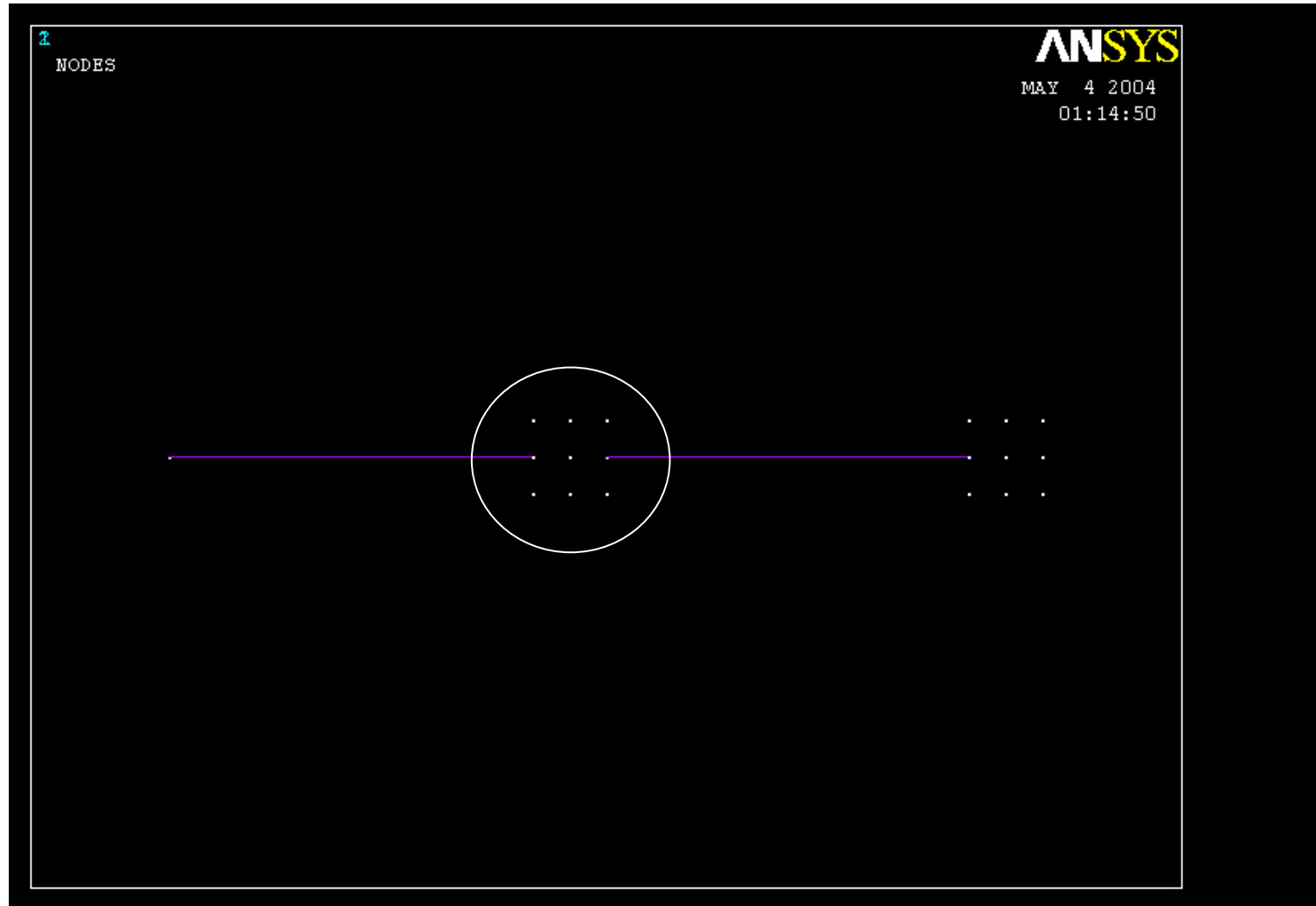
# Example – Define Loads



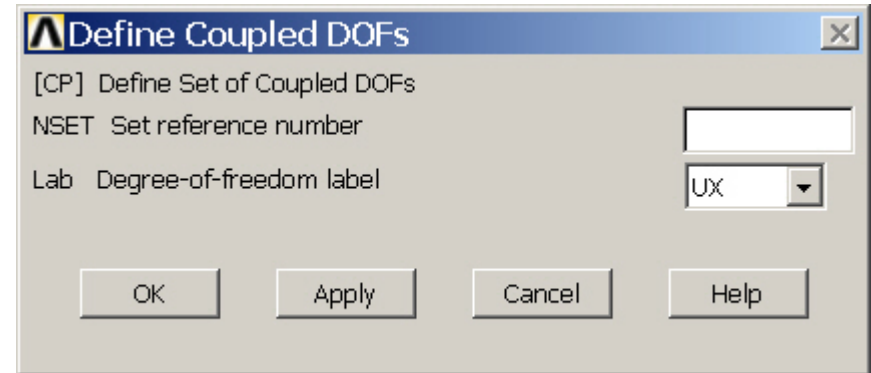
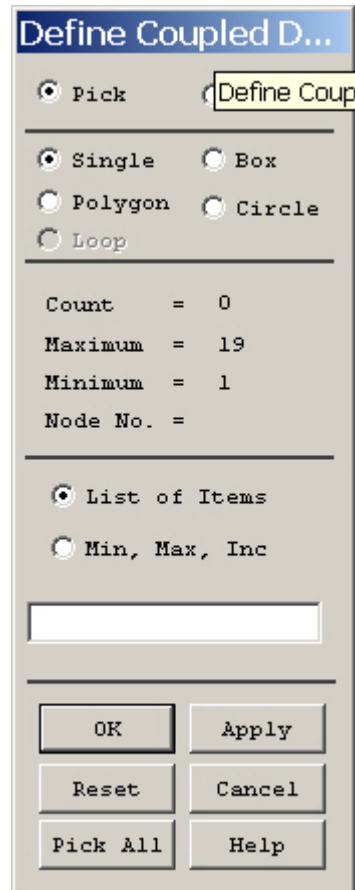
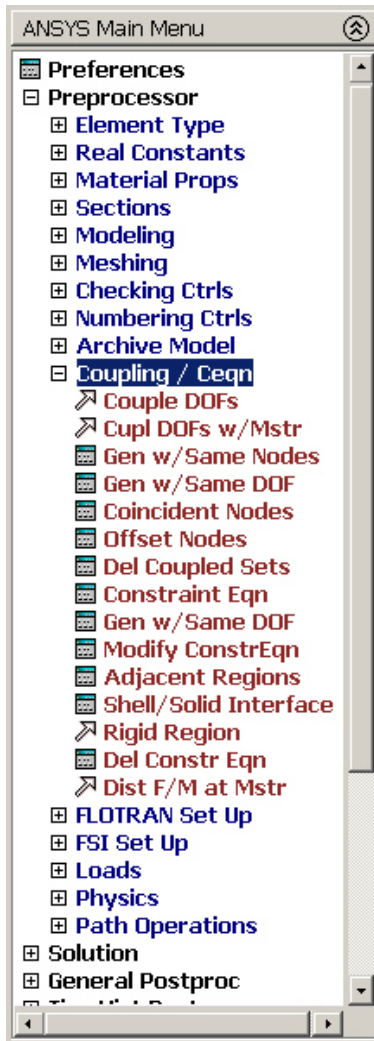
# Example – Coupling DOFs



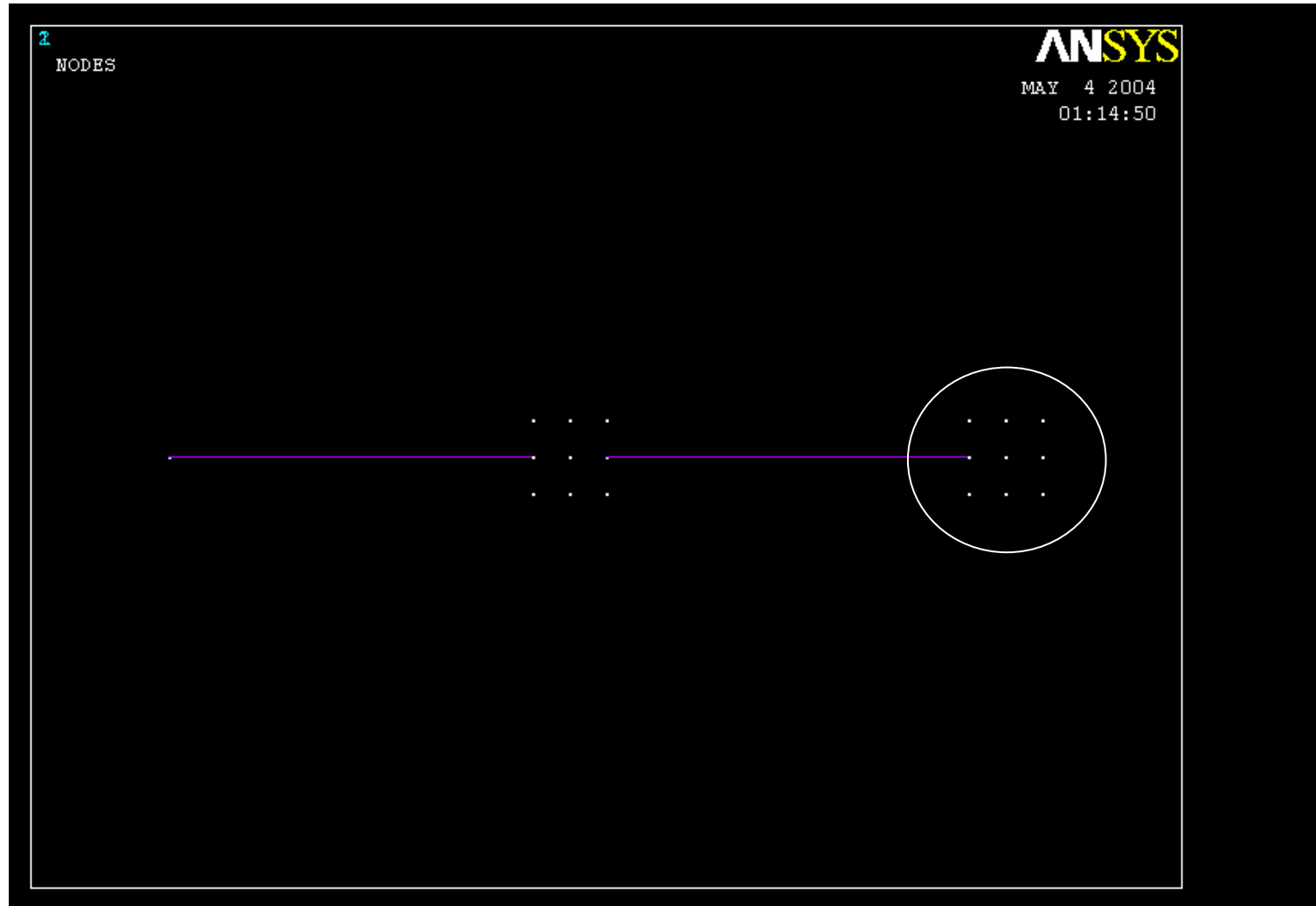
# Example – Coupling DOFs



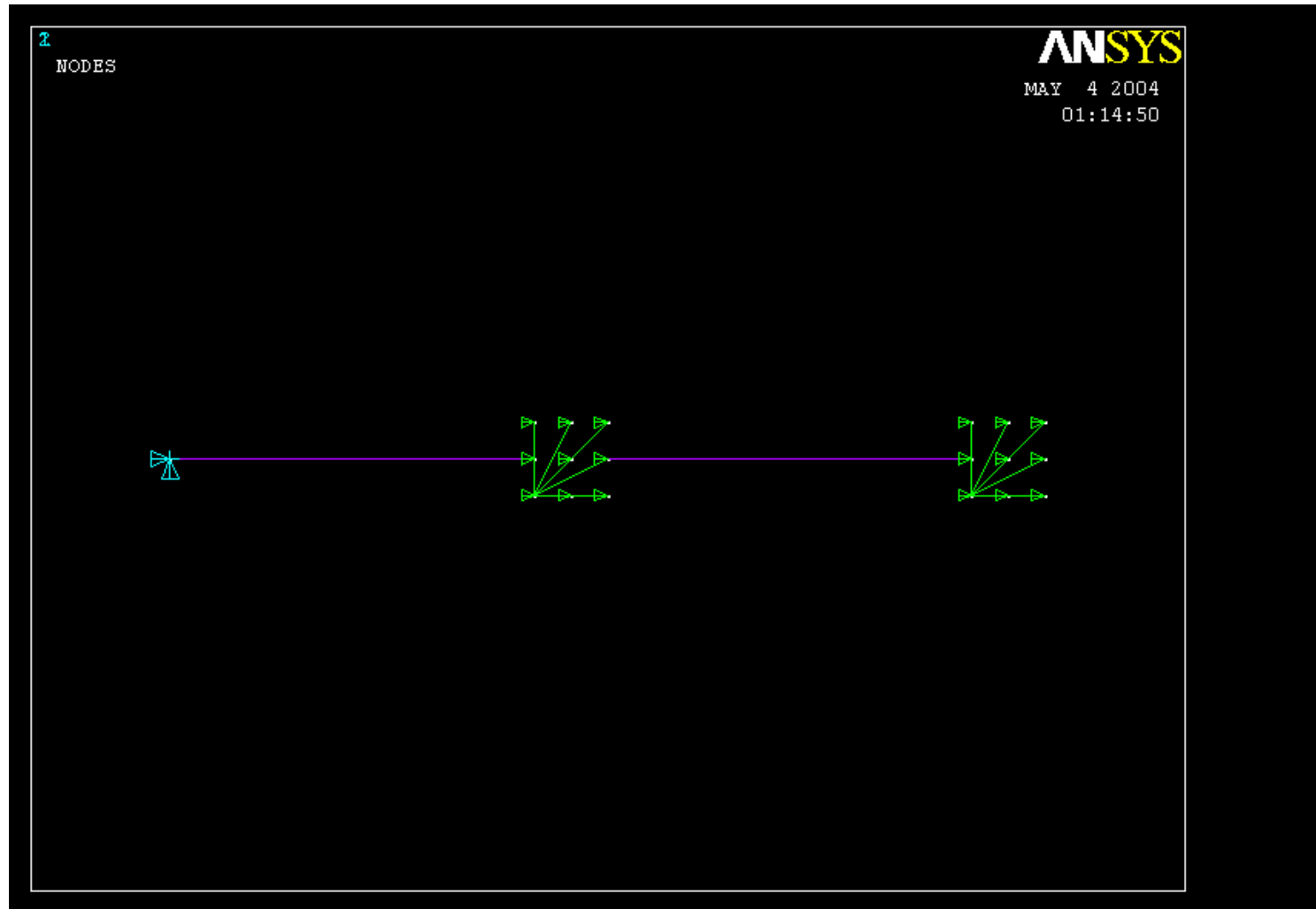
# Example – Coupling DOFs



# Example – Coupling DOFs

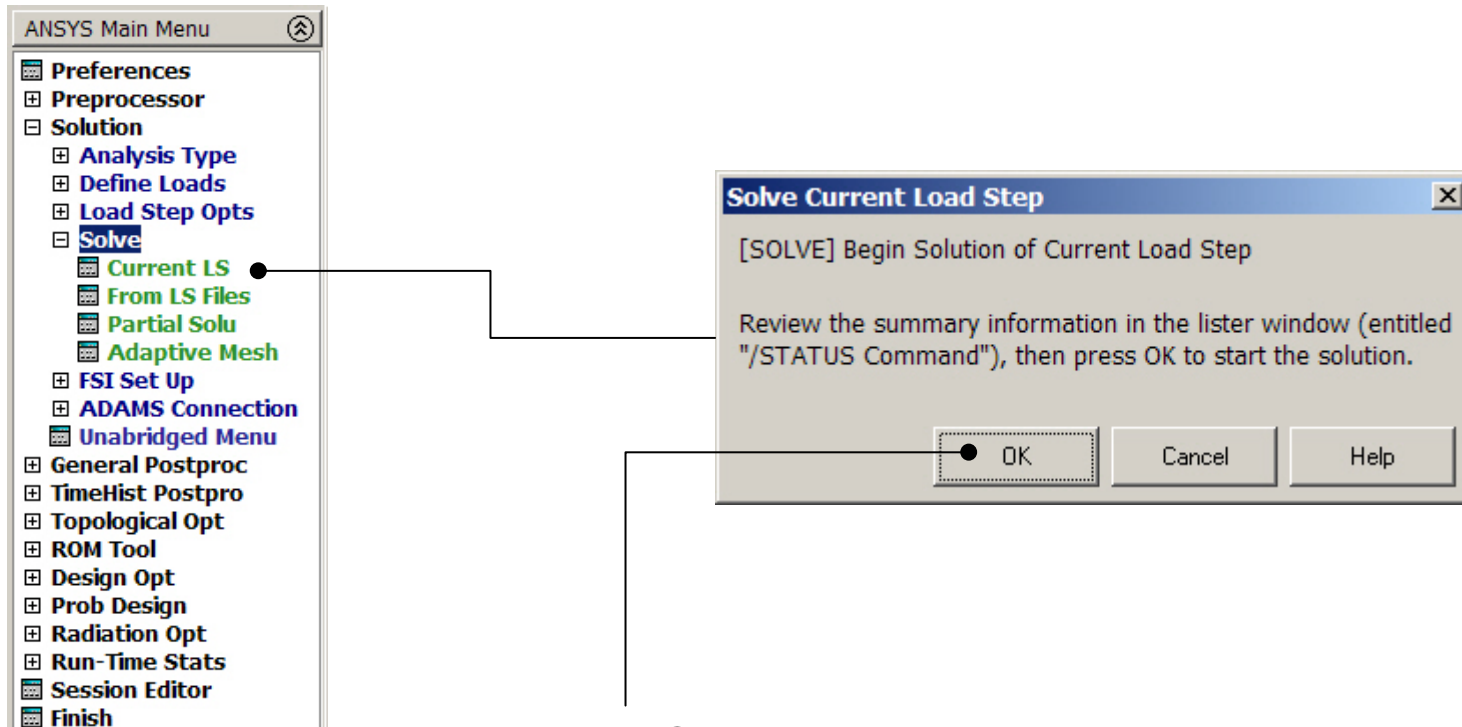


# Example – Analysis model



# Example - Solve

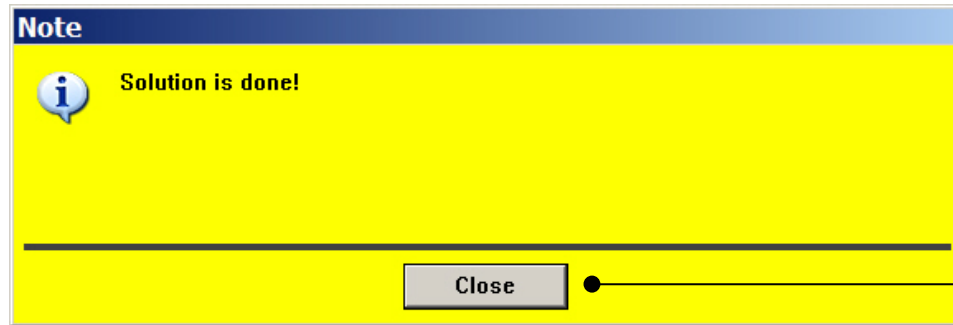
**Solution > Solve > Current LS**



Press OK

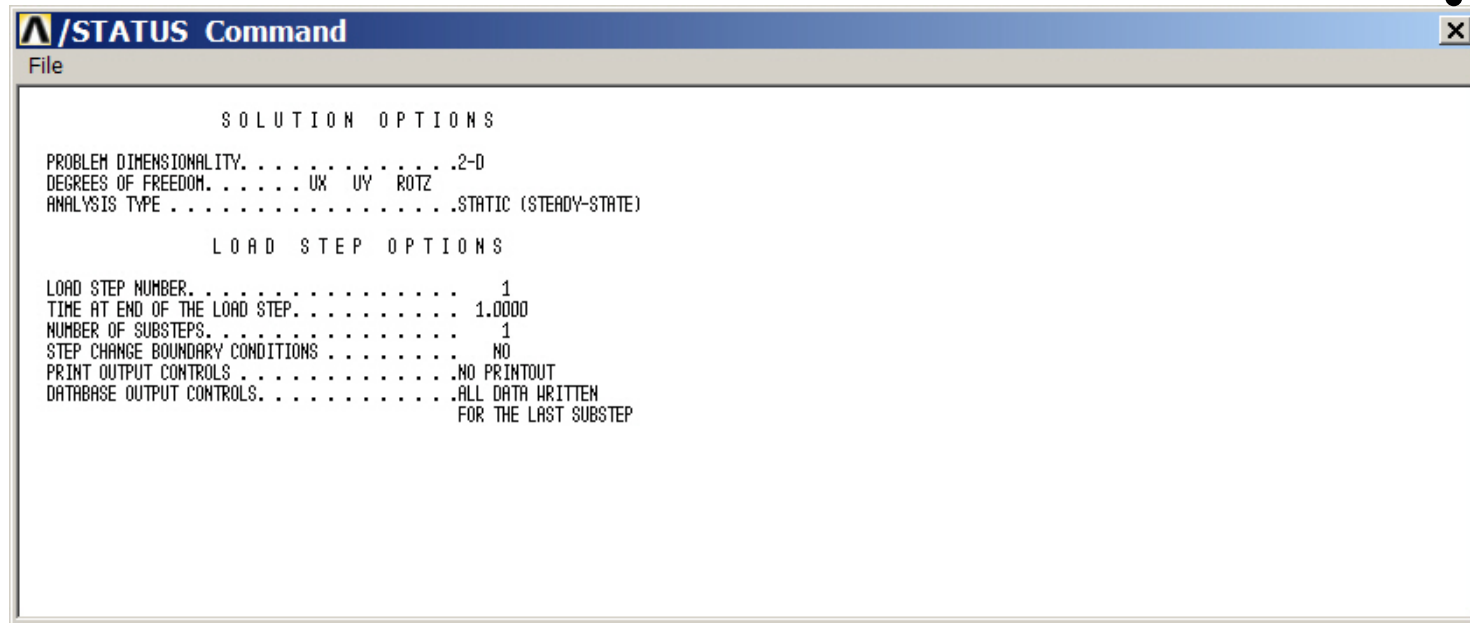
Example0430

# Example - Solve



Press Close

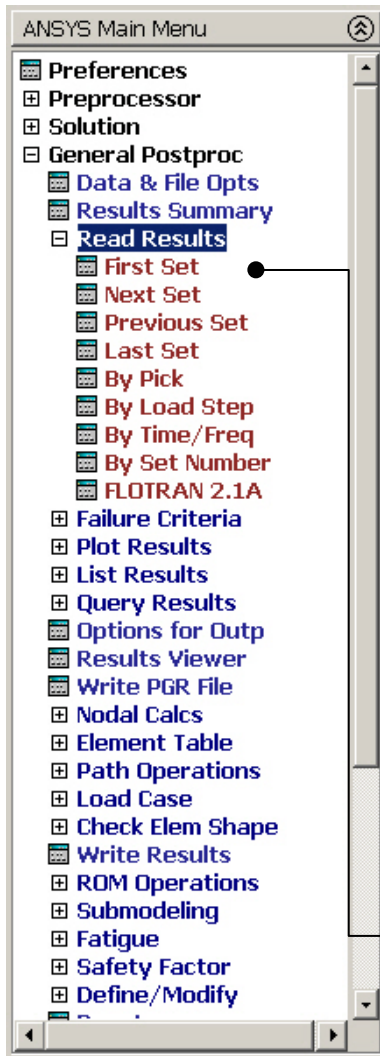
Press here  
to Close



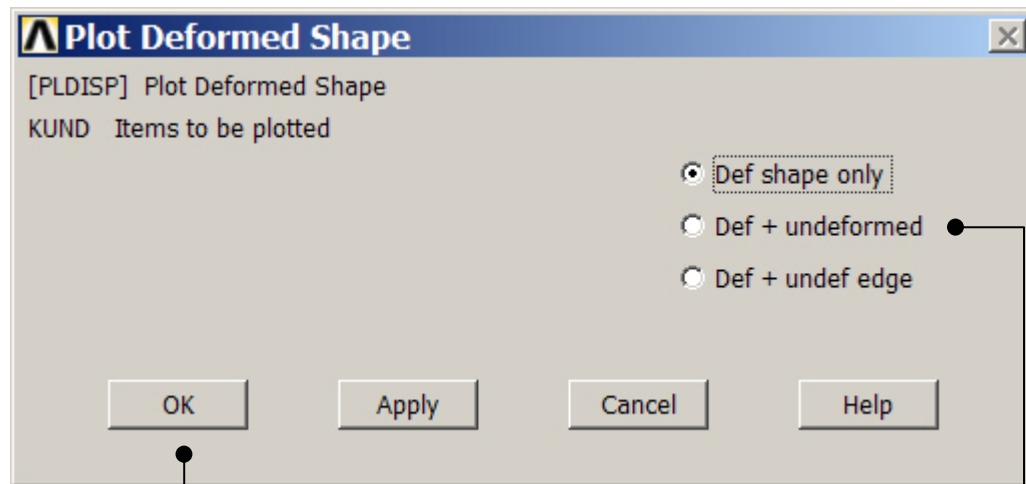


# Example – Read Results

General Postproc > Plot Results > Deformed Shape

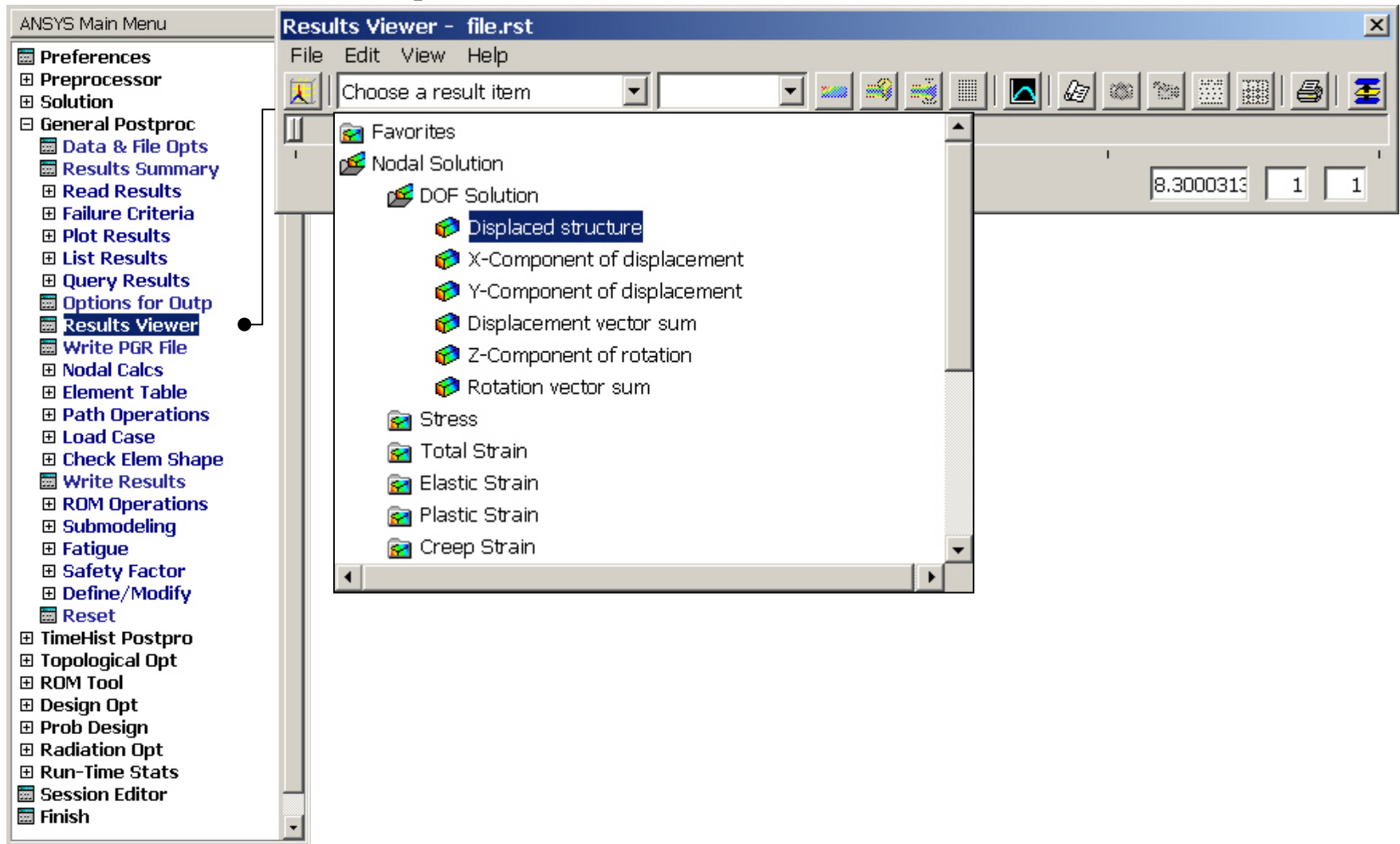


Read first set

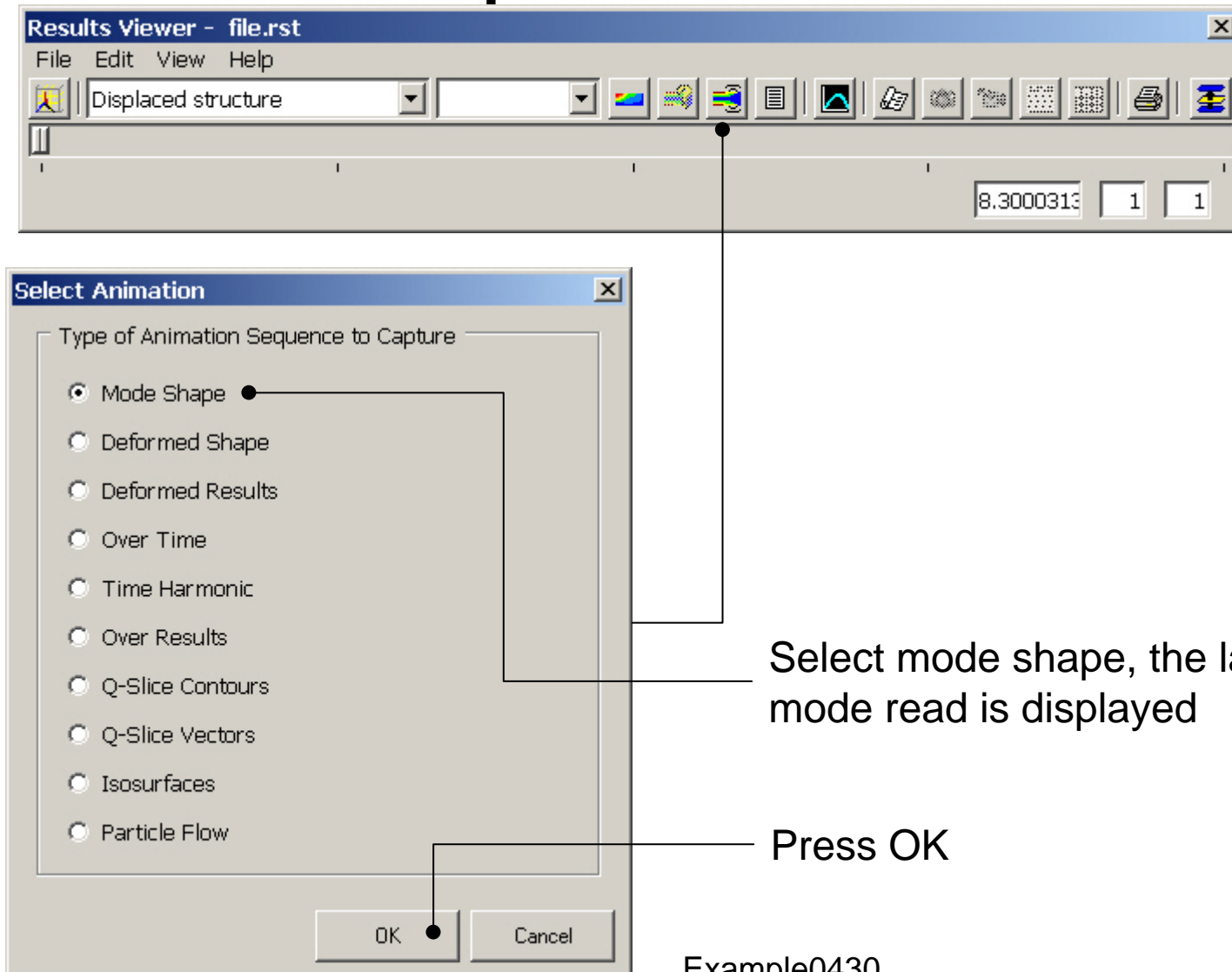


Select "Def+undeformed"  
and Press OK

# Example – Result viewer

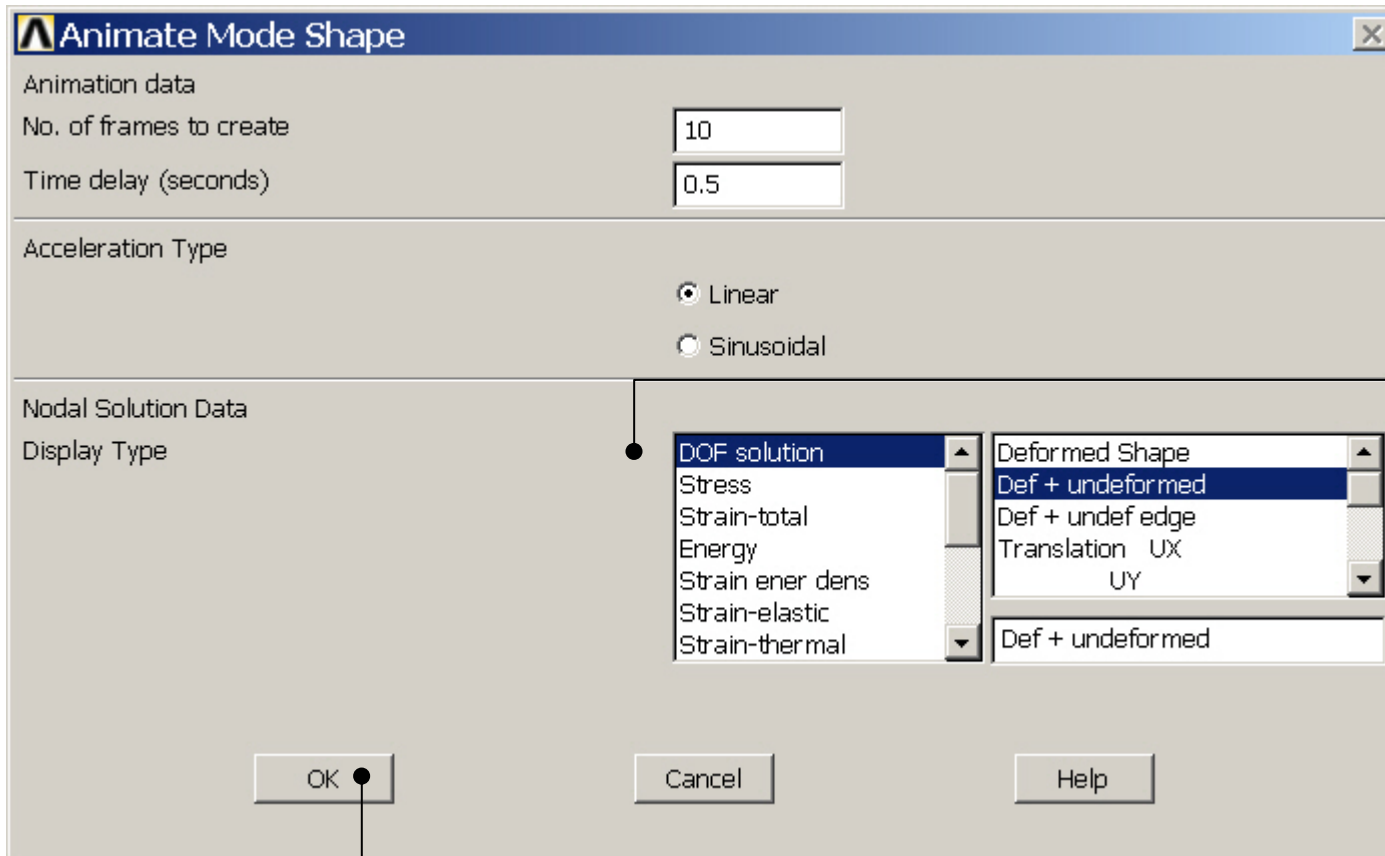


# Example – Result viewer

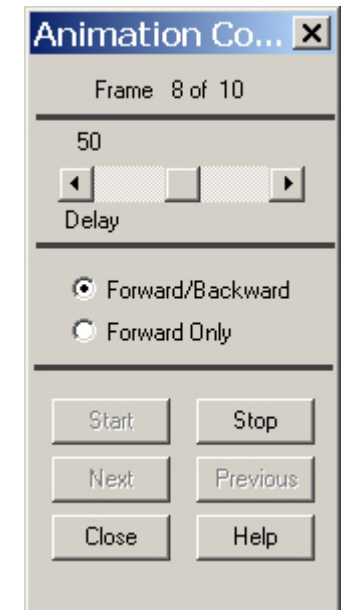


Example0430

# Example – Result viewer



Select DOF solution and Def+undeformed



Press OK and animation is controlled by the following dialog

# Example – Report generator

Modal analysis