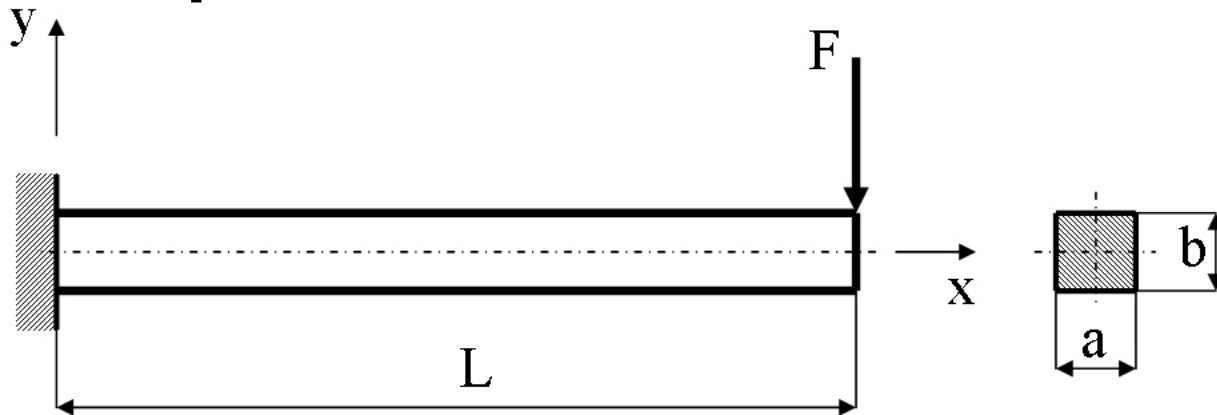# Course in ANSYS

Example0200

Example – Cantilever beam

Objective:

Compute the maximum deflection and locate
point of maximum deflection

Tasks:

How should this be modelled?

Compare results with results obtained
from beam theory?

Topics:

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

$E = 210000 \text{N/mm}^2$
$\nu = 0.3$
$L = 100 \text{mm}$
$a = 10 \text{mm}$
$b = 10 \text{mm}$
$F = 100 \text{N}$

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.)
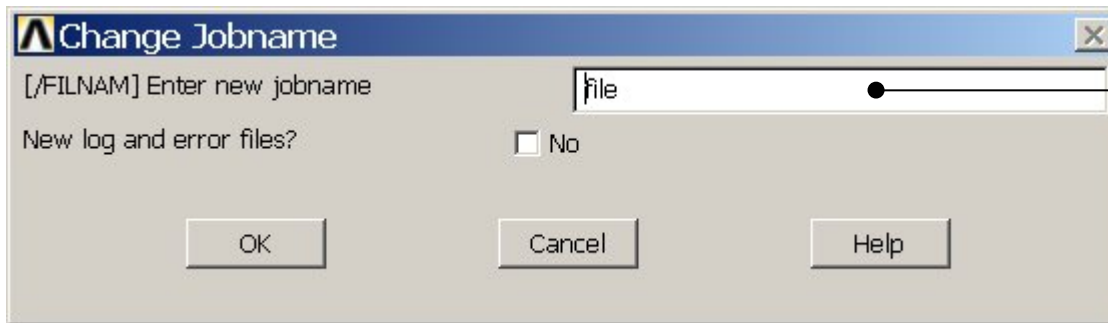- How Much Detail to Include
- Appropriate Mesh Density

Example - title

Utility Menu > File > Change Jobname •————————— GUI
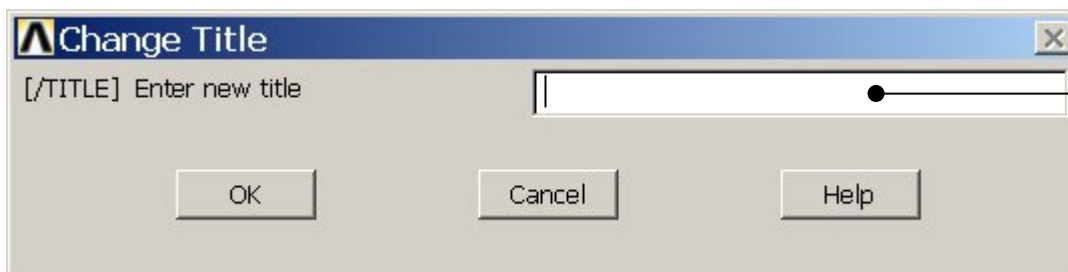/jobname, Example0200 •——————————————— Command line entry

Enter: Example0200

Change Jobname		×
[/FILNAM] Enter new jobname	file •	
New log and error files?	☐ No	

| OK | Cancel | Help |

Utility Menu > File > Change Title   Enter: Cantilever beam
/title, Cantilever beam

Change Title	×
[/TITLE] Enter new title	| •

| OK | Cancel | Help |

Example - Keypoints

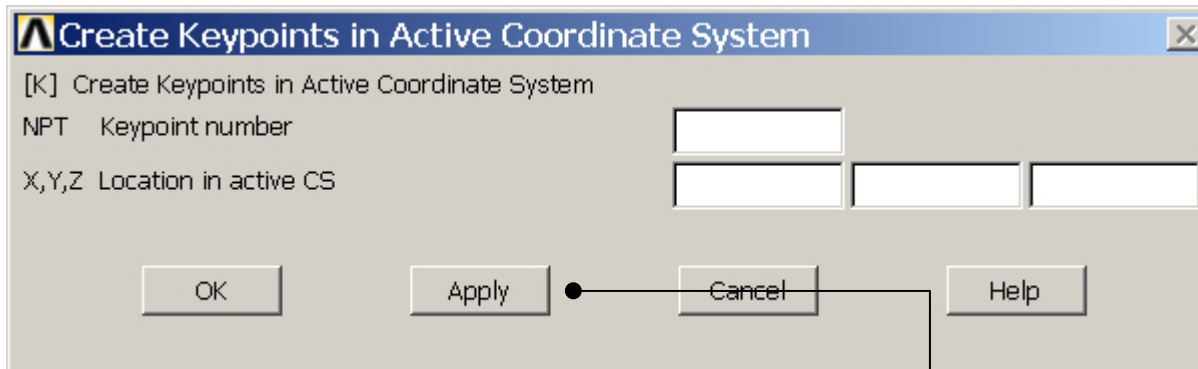**Preprocessor > Modeling > Create > Keypoints > In Active CS**

/PREP7
K,,,,
K,,100,,
K,,100,10,
K,,,10,

General format:
K,#,X,Y,Z

\# Keypoint number
X Keypoint x-coordinate
Y Keypoint y-coordinate
Z Keypoint z-coordinate

Create Keypoints in Active Coordinate System ☒

[K] Create Keypoints in Active Coordinate System

NPT   Keypoint number   [_____]

X,Y,Z Location in active CS   [_____] [_____] [_____]

OK   Apply   Cancel   Help

Press **Apply** for KP1
Enter 100 in the first field and
Press **Apply** for KP2
Enter 100 in the first field and Enter 10 in the second field and
Press **Apply** for KP3
Enter 10 in the second field and
Press **Apply** for KP4

Example - Lines

Preprocessor > Modeling > Create > Lines > Lines > Straight Line
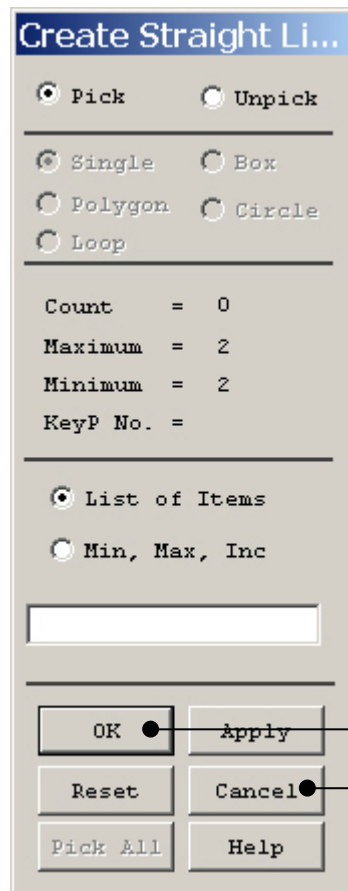
Create a line between Keypoint 1 and Keypoint 2 and so on.

L,1,2
L,2,3
L,3,4
L,4,1

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:
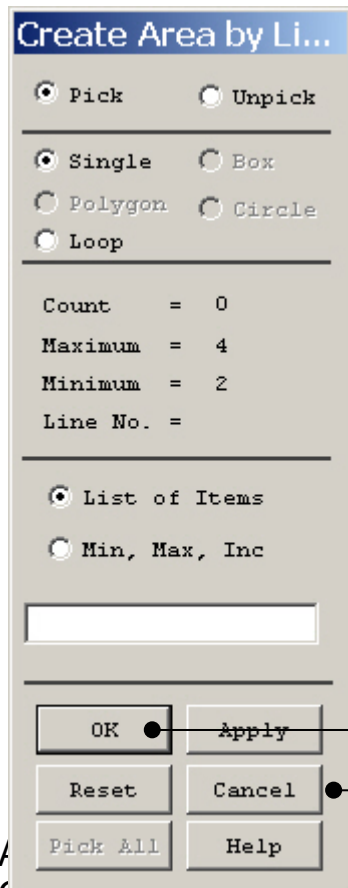
Pick: upward arrow

Unpick: downward arrow

Press OK or Cancel to finish selection

Create Straight Li...

- ⦿ Pick   ○ Unpick
- ⦿ Single   ○ Box
- ○ Polygon ○ Circle
- ○ Loop

Count   =   0
Maximum =   2
Minimum =   2
KeyP No. =

- ⦿ List of Items
- ○ Min, Max, Inc

OK	Apply
Reset	Cancel
Pick All	Help

Example - Areas

Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines
Create an area bounded by line 1, 2, 3, and 4.
AL,1,2,3,4

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:
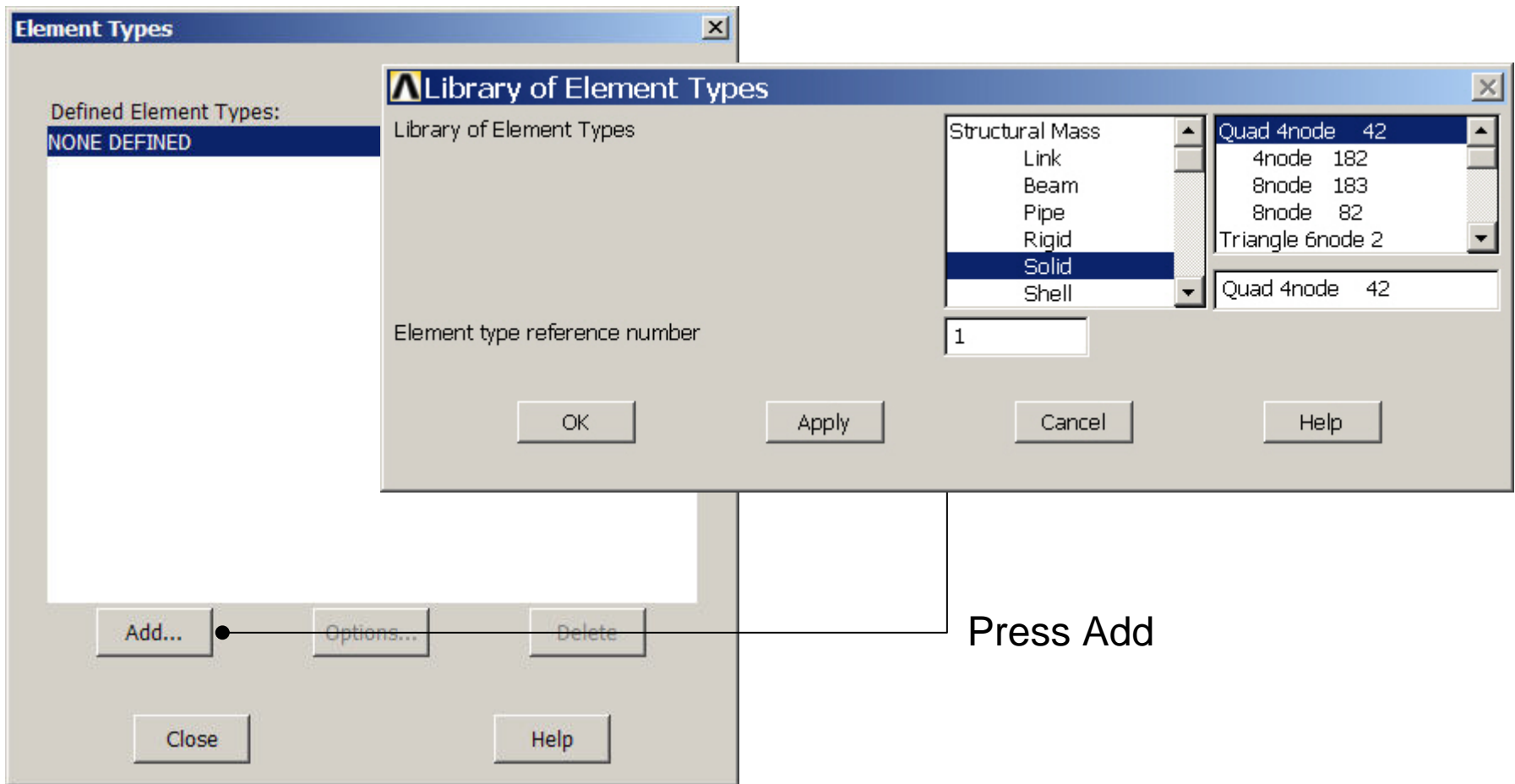
Pick: upward arrow

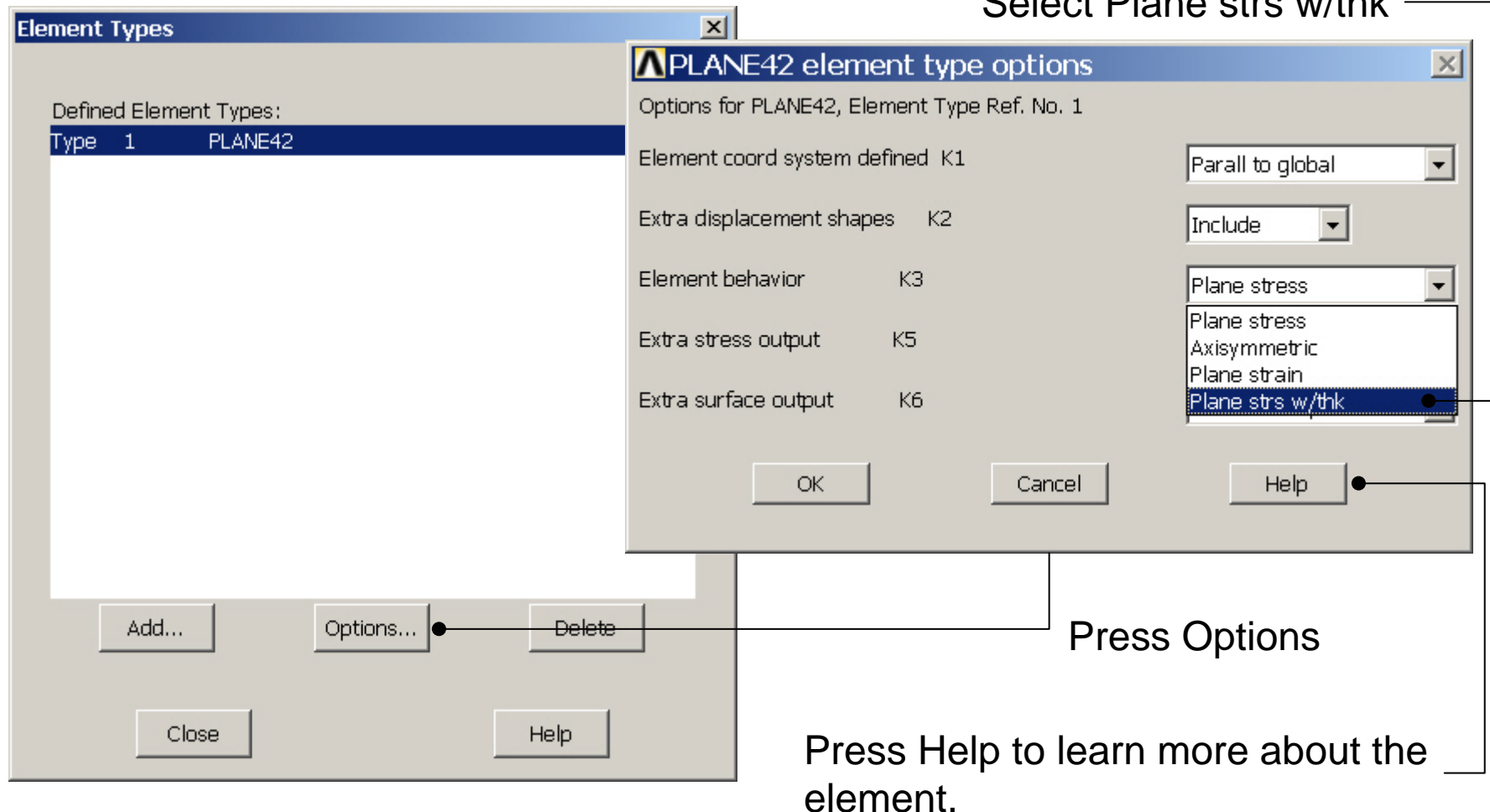Unpick: downward arrow

Press OK or Cancel to finish selection

Create Area by Li...

- ● Pick   ○ Unpick
- ● Single   ○ Box
- ○ Polygon   ○ Circle
- ○ Loop

Count = 0
Maximum = 4
Minimum = 2
Line No. =

- ● List of Items
- ○ Min, Max, Inc

OK	Apply
Reset	Cancel
Pick All	Help

Example0200

8

Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete

Press Add

Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete

Select Plane strs w/thk

Element Types

Defined Element Types:

Type   1   PLANE42

Add...   Options...   Delete

Close   Help

PLANE42 element type options

Options for PLANE42, Element Type Ref. No. 1

Element coord system defined K1	Parall to global
Extra displacement shapes K2	Include
Element behavior K3	Plane stress
Extra stress output K5	Plane stress
	Axisymmetric
	Plane strain
Extra surface output K6	Plane strs w/thk

OK   Cancel   Help

Press Options

Press Help to learn more about the element.
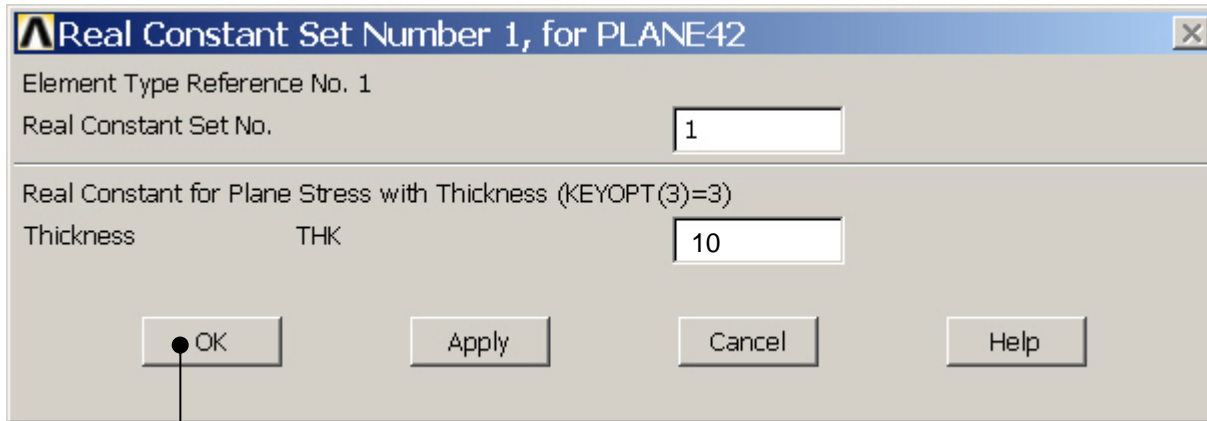
Example – Real Constants

Preprocessor > Real Constants > Add
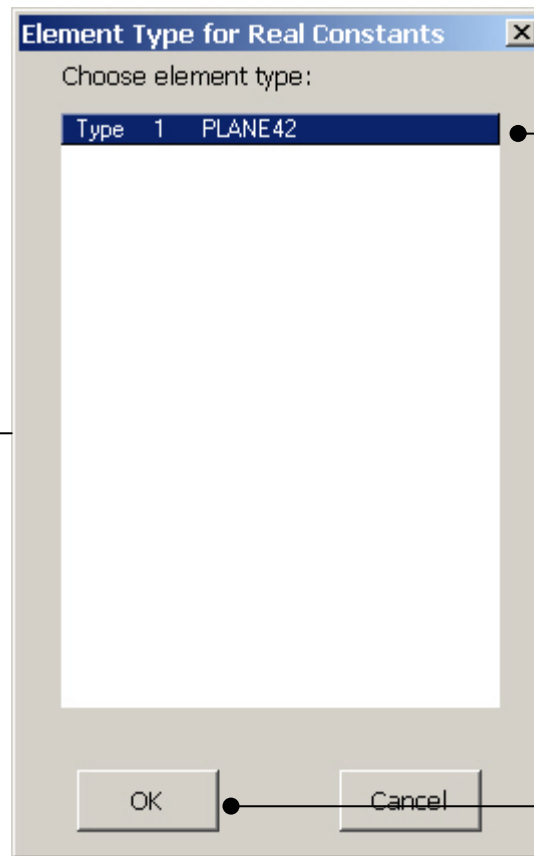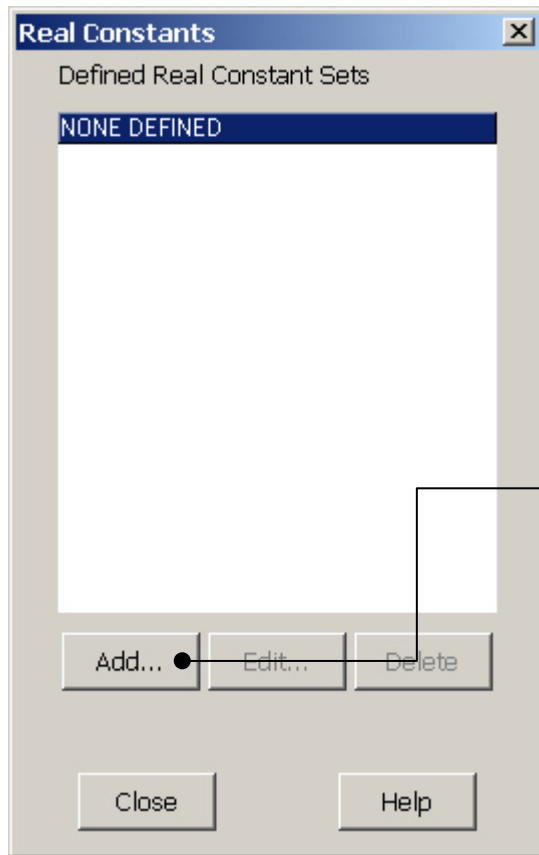
Place the cursor on the relevant element and press OK

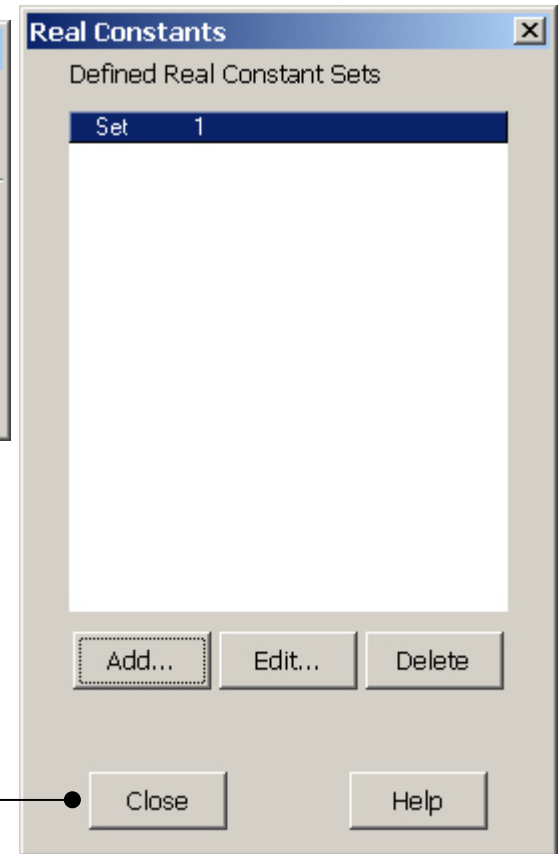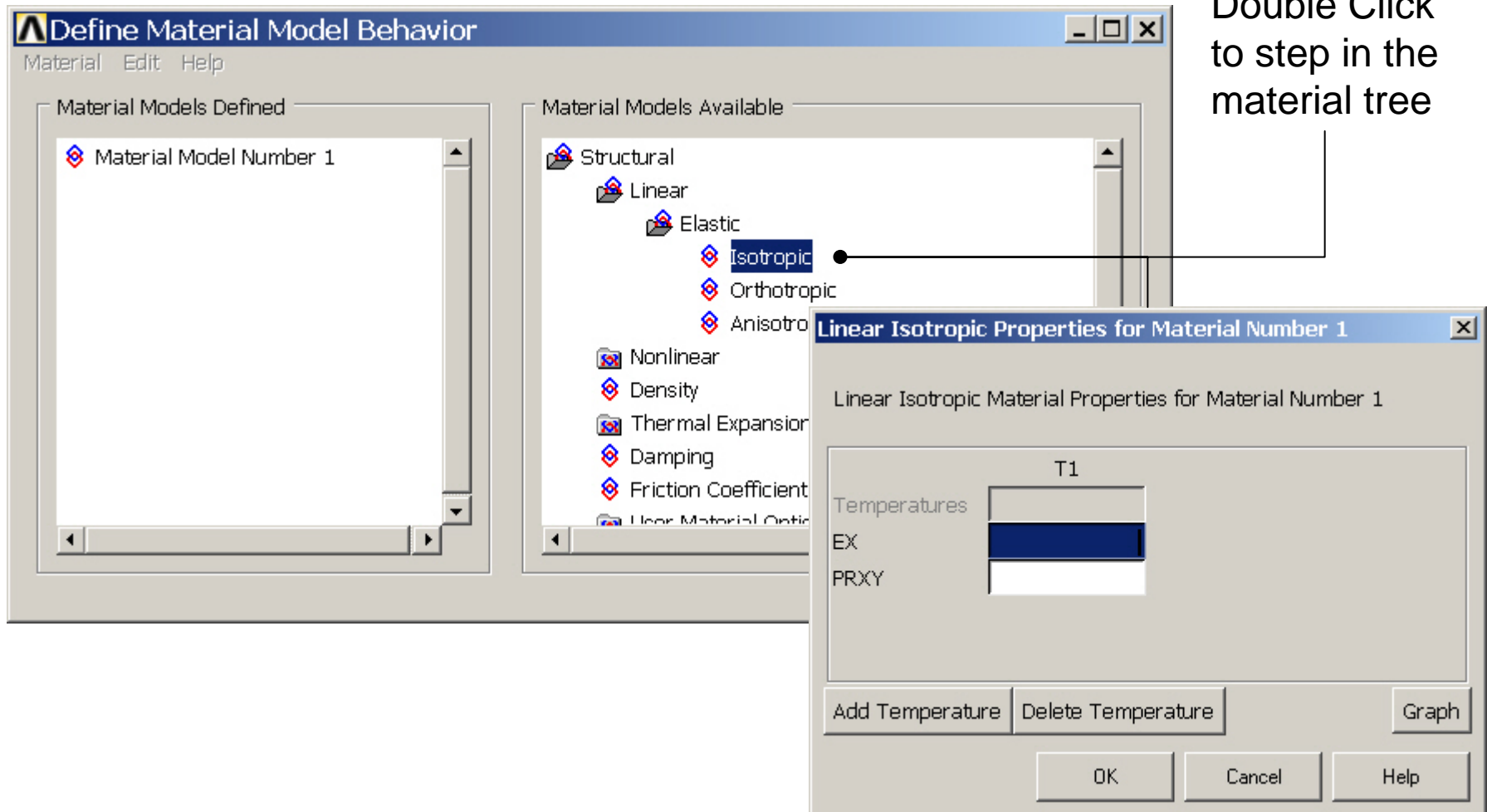# Example - Real Constants
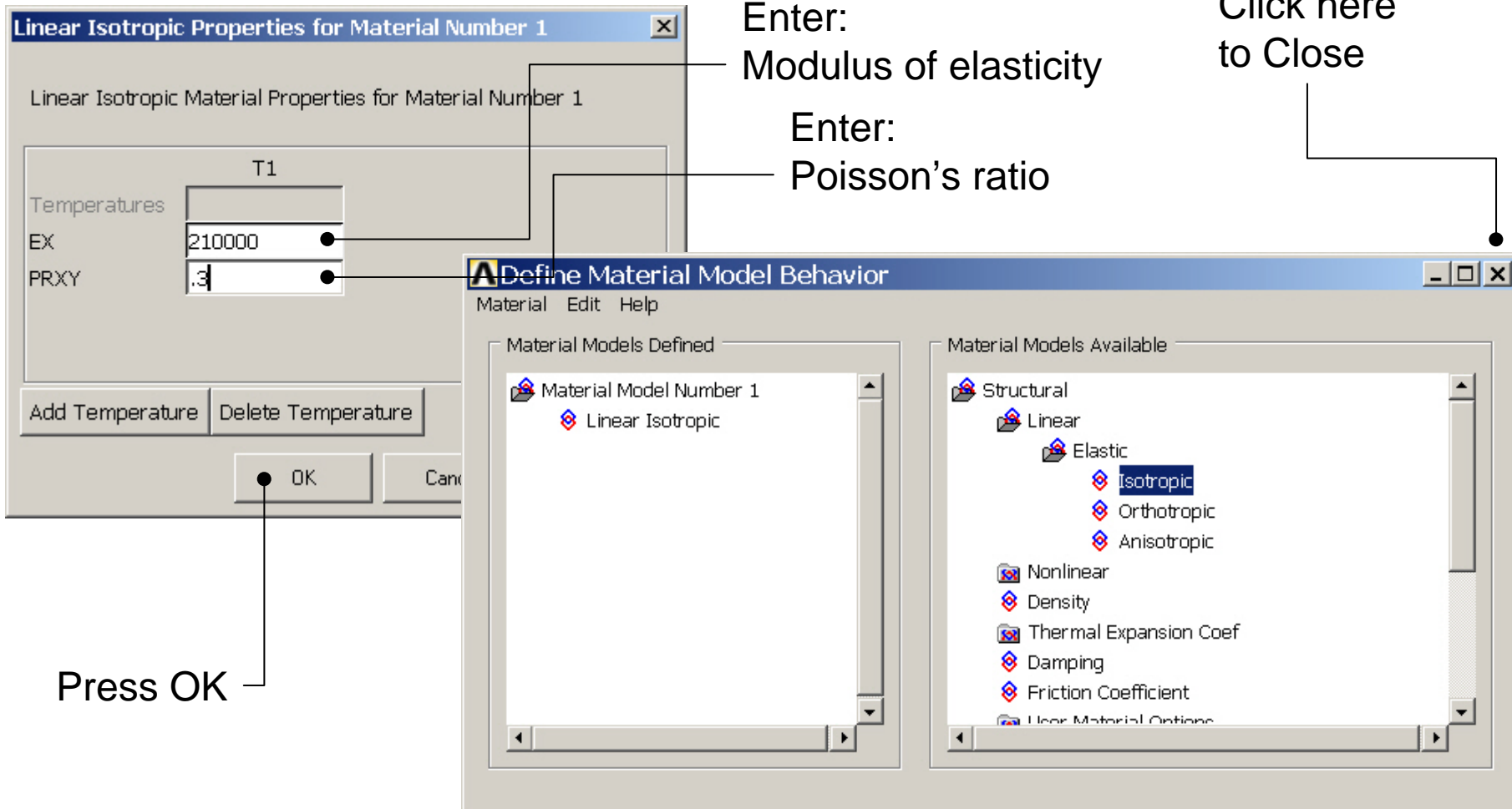
Preprocessor > Real Constants > Add

Real Constant Set Number 1, for PLANE42

Element Type Reference No. 1

Real Constant Set No.   `1`

Real Constant for Plane Stress with Thickness (KEYOPT(3)=3)

Thickness   THK   `10`

| OK | Apply | Cancel | Help |

Real Constants

Defined Real Constant Sets

Set   1

| Add... | Edit... | Delete |

Press Close
to finish

| Close | Help |

Press OK

Example - Material Properties

Preprocessor > Material Props > Material Models

Double Click to step in the material tree

Example - Material Properties

Preprocessor > Material Props > Material Models

Enter:
Modulus of elasticity

Enter:
Poisson's ratio

Click here
to Close

Press OK

Example - Meshing

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines
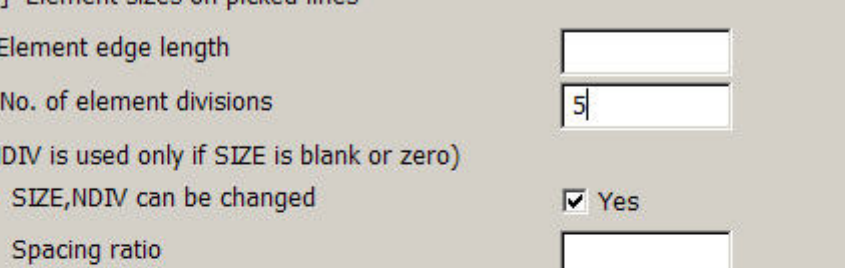
Select/Pick Lines to specify mesh size for

Pick the two longest lines

Press OK when finish with selection

Example - Meshing

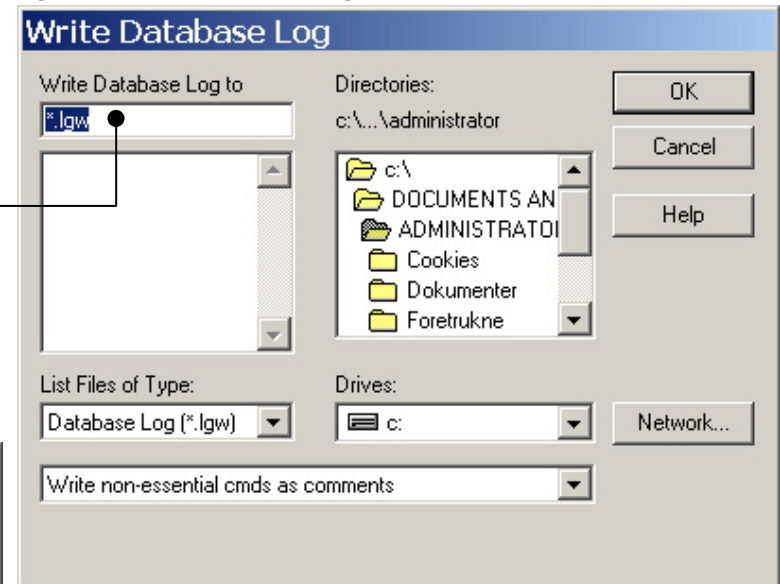**Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided**
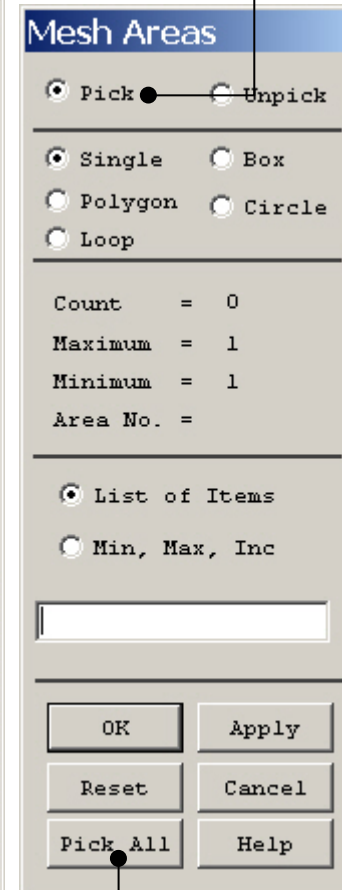
Select individual areas to be meshed

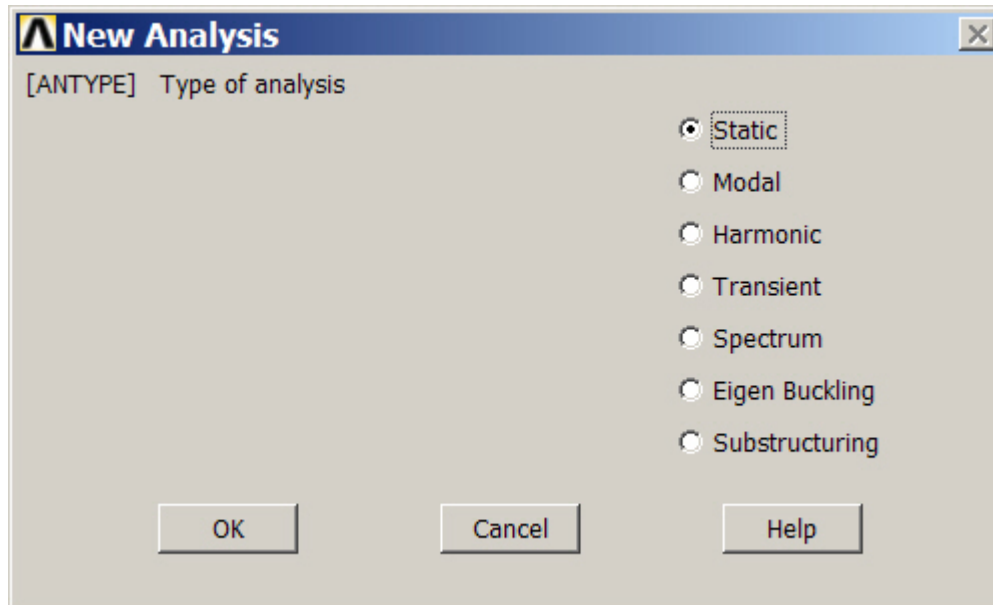**NB**: It is often necessary to "Clear" the model for example if Element Type or model geometry is to be changed

Select all areas defined to be meshed

Example – Analysis Type

File > Write DB log file
Enter "example0200.lgw"

Write Database Log

Write Database Log to

`*.lgw`

Directories:
c:\...\administrator

- c:\
  - DOCUMENTS AN
  - ADMINISTRATOI
    - Cookies
    - Dokumenter
    - Foretrukne

List Files of Type:
Database Log (*.lgw)

Drives:
c:

Write non-essential cmds as comments

OK | Cancel | Help | Network...

Solution > Analysis Type > New Analysis

New Analysis

[ANTYPE]   Type of analysis

- ● Static
- ○ Modal
- ○ Harmonic
- ○ Transient
- ○ Spectrum
- ○ Eigen Buckling
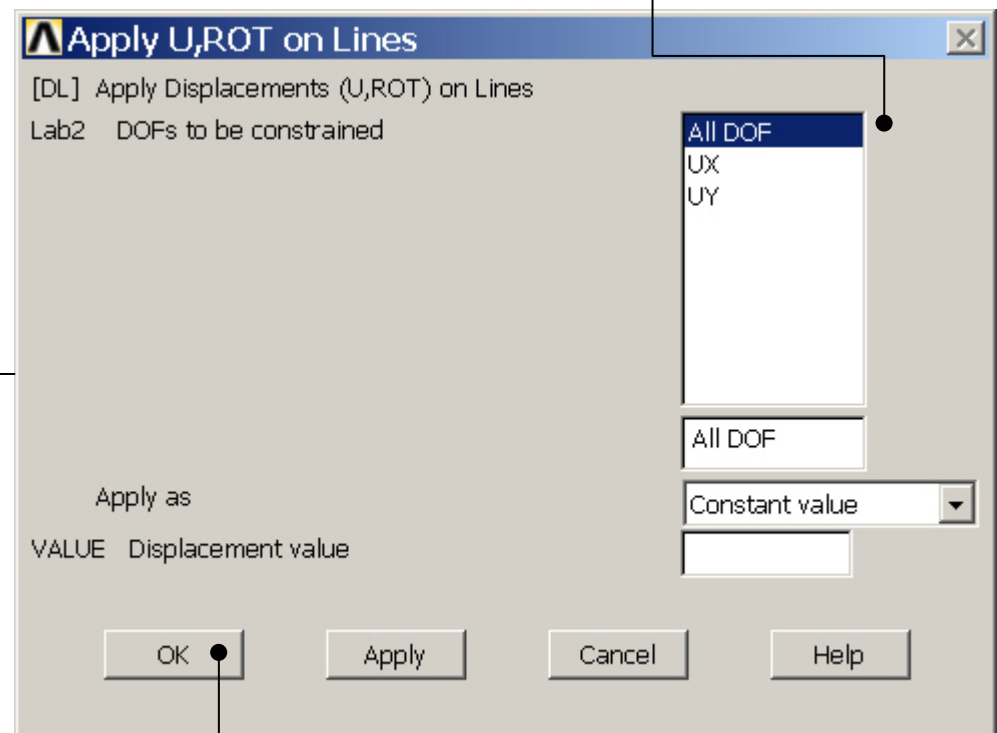- ○ Substructuring

OK | Cancel | Help

Example – Define Loads

Solution > Define Loads > Apply > Structural > Displacement > On Lines
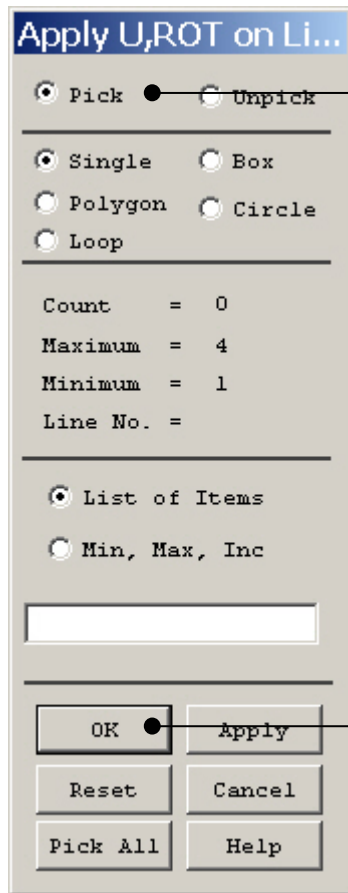
Select Line 4

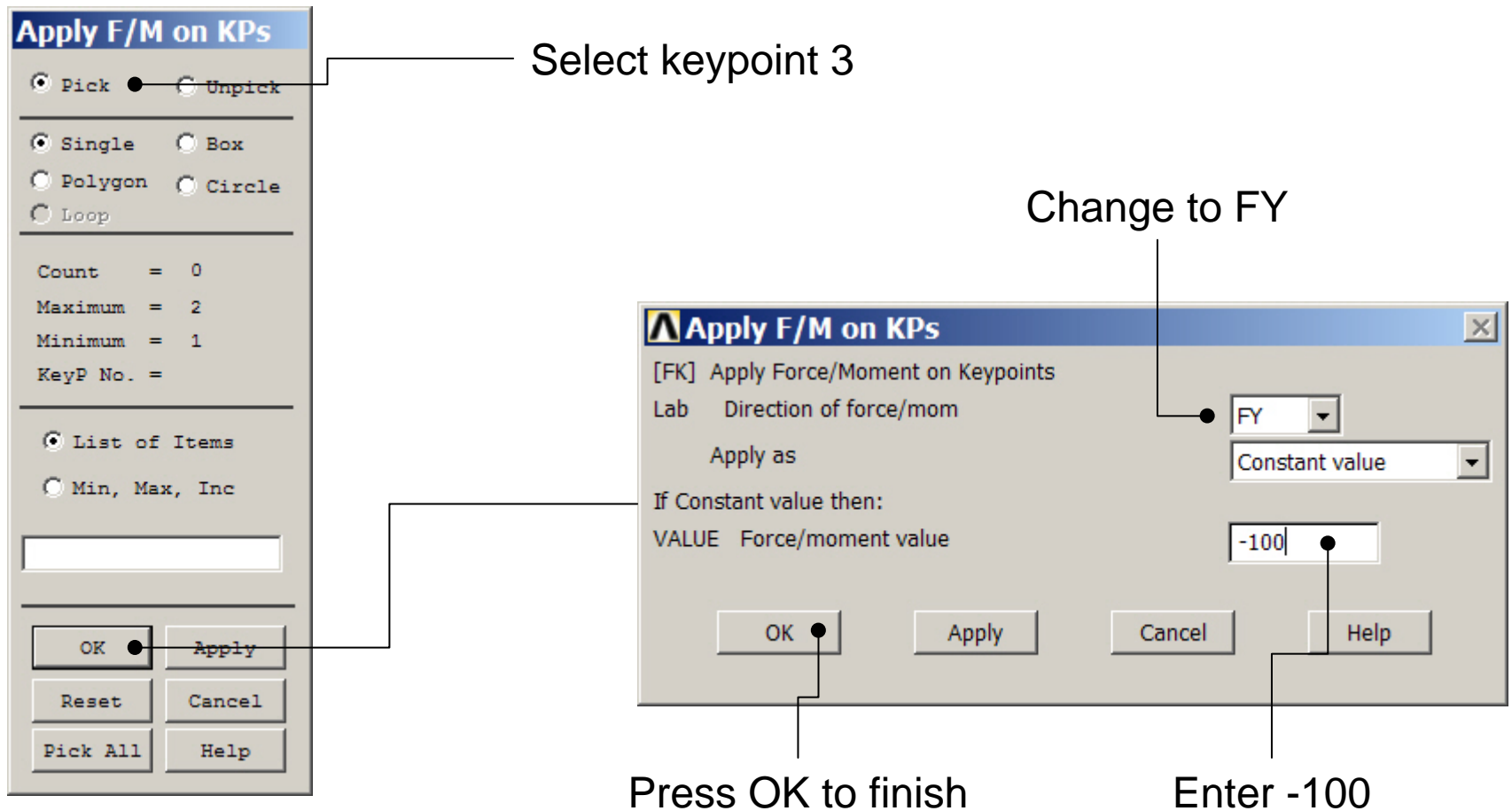Select All DOF to fix/clamp the beam
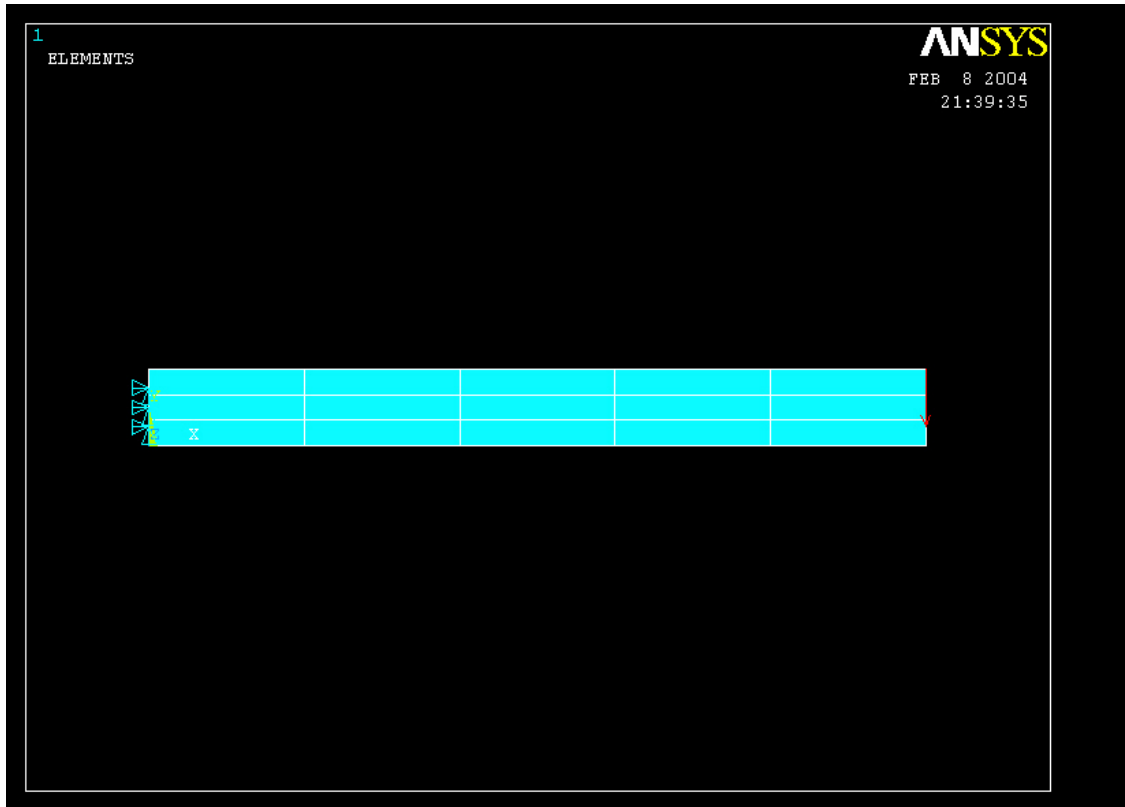
Press OK

Example – Define Loads

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

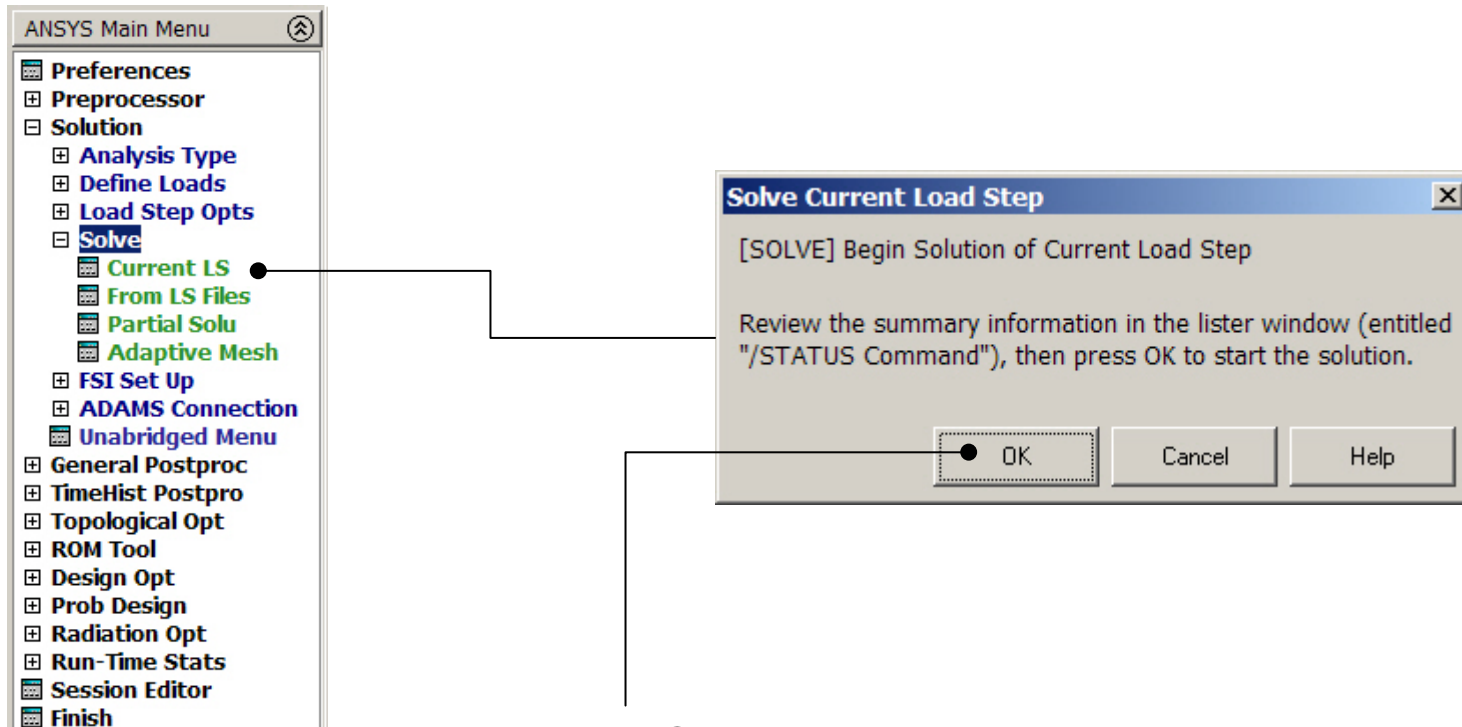Select keypoint 3

Change to FY

Press OK to finish

Enter -100

Example - Save

Display of Analysis model
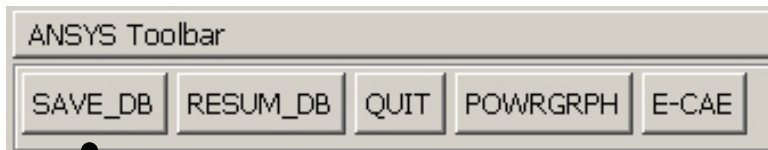
Save the model

Example - Solve

Solution > Solve > Current LS

Press OK

Example0200

Computational Mechanics, AAU, Esbjerg

21

Example - Solve
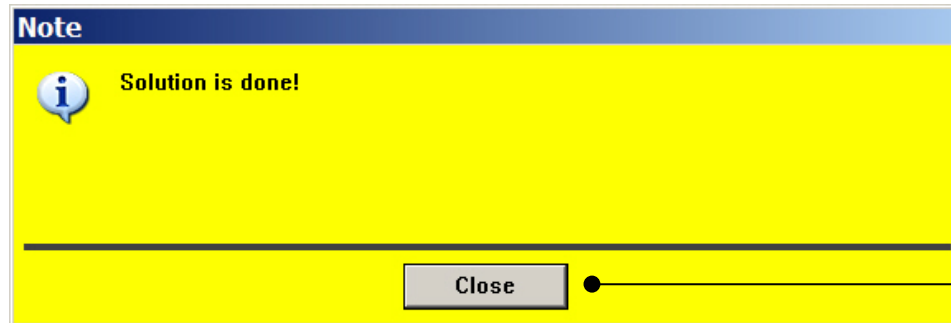
Press Close

Press here
to Close

Note

ℹ **Solution is done!**

Close

/STATUS Command

File

```
        SOLUTION  OPTIONS

PROBLEM DIMENSIONALITY. . . . . . . . . . . .2-D
DEGREES OF FREEDOM. . . . . . UX  UY  ROTZ
ANALYSIS TYPE . . . . . . . . . . . . . . . .STATIC (STEADY-STATE)

        LOAD  STEP  OPTIONS

LOAD STEP NUMBER. . . . . . . . . . . . . . .     1
TIME AT END OF THE LOAD STEP. . . . . . . . . 1.0000
NUMBER OF SUBSTEPS. . . . . . . . . . . . . .     1
STEP CHANGE BOUNDARY CONDITIONS . . . . . . .    NO
PRINT OUTPUT CONTROLS . . . . . . . . . . . .NO PRINTOUT
DATABASE OUTPUT CONTROLS. . . . . . . . . . .ALL DATA WRITTEN
                                            FOR THE LAST SUBSTEP
```
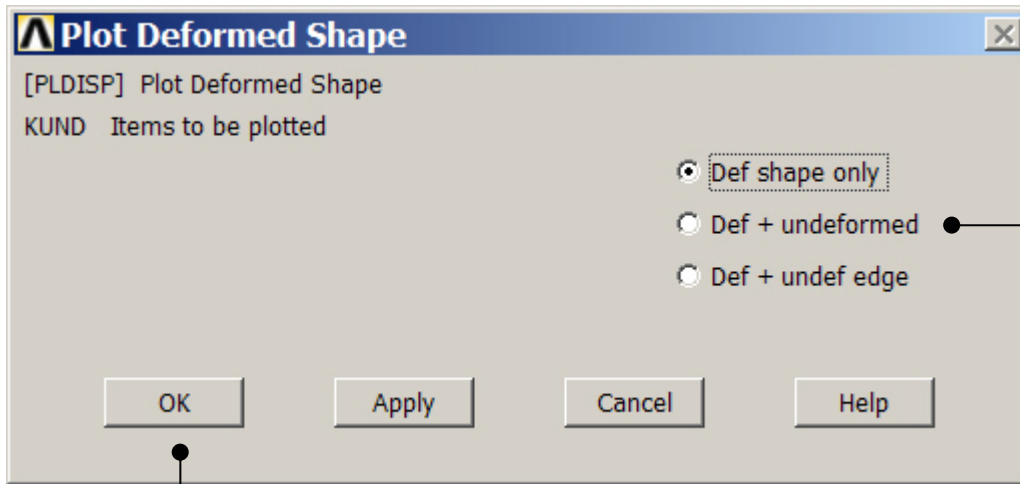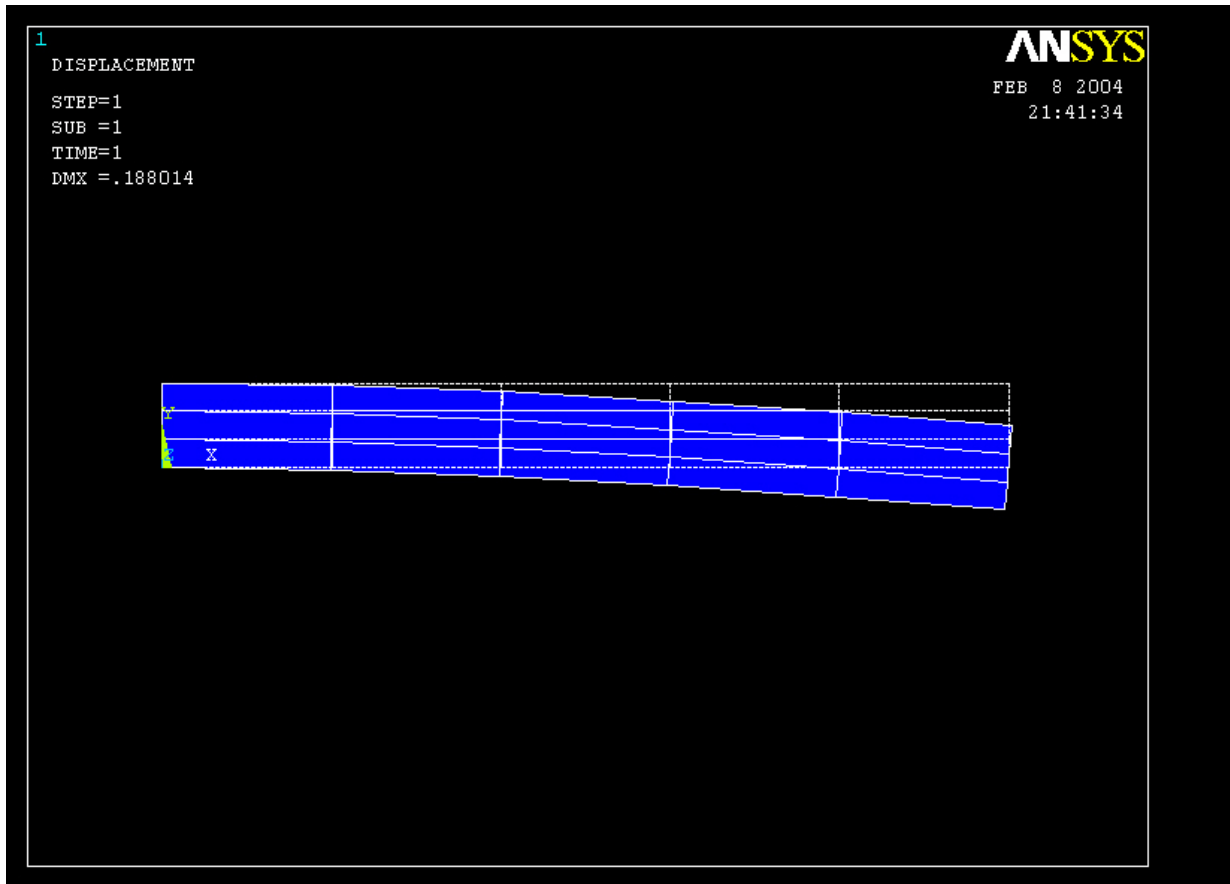
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape

Select "Def+undeformed"
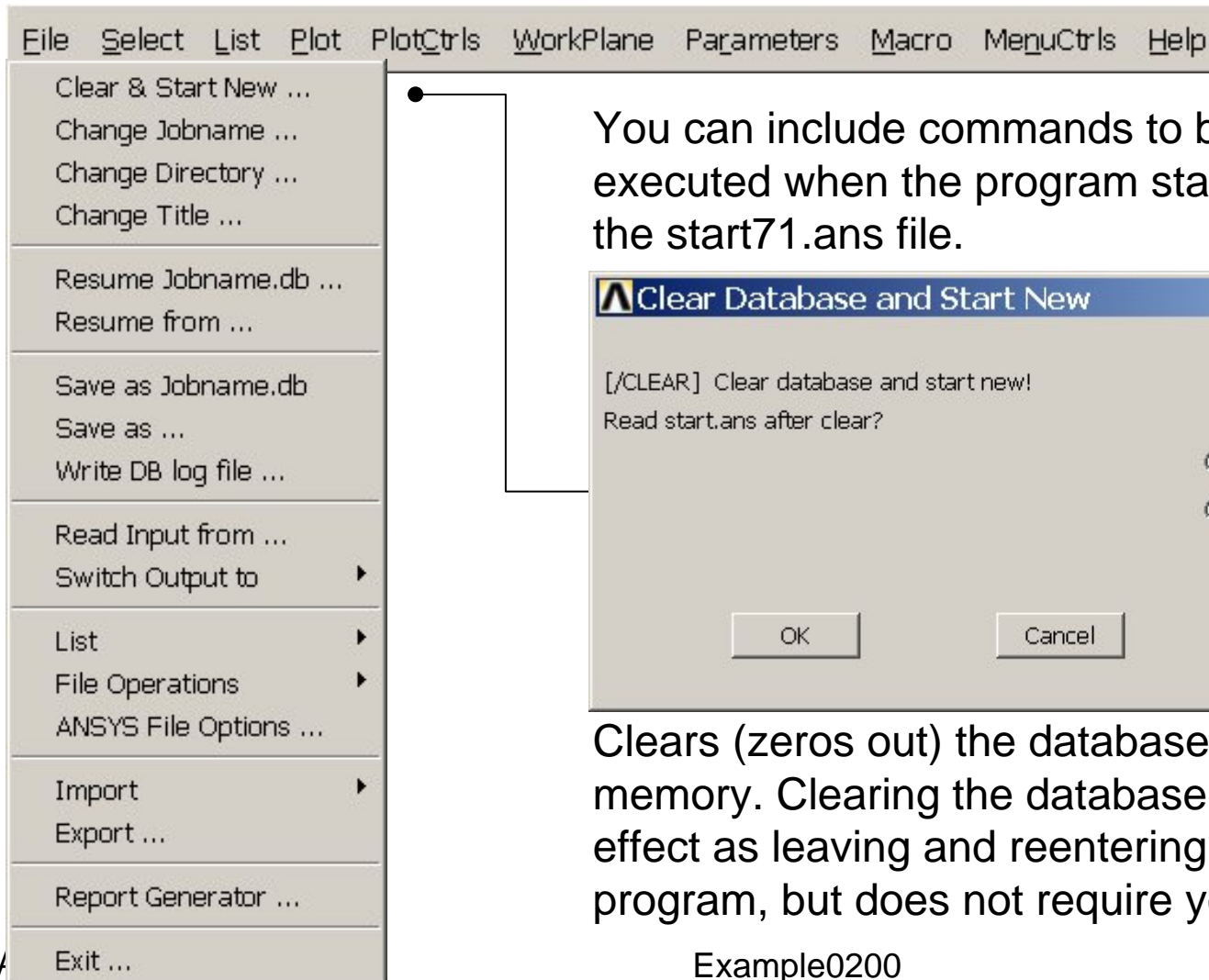and Press OK

Example - PostProcessing

Read Maximum displacement: DMX

Example – Comments/Questions

- What did change compared to the Beam model?

- The "example0200.lgw" can be edited in "Notepad"

- What are the assumptions in this case?

- Will the shape or the number of elements affect the solution?

File menu

File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help

Clear & Start New ...
Change Jobname ...
Change Directory ...
Change Title ...

Resume Jobname.db ...
Resume from ...

Save as Jobname.db
Save as ...
Write DB log file ...

Read Input from ...
Switch Output to ▸

List ▸
File Operations ▸
ANSYS File Options ...

Import ▸
Export ...

Report Generator ...

Exit ...

You can include commands to be executed when the program starts up in the start71.ans file.

Clear Database and Start New ×

[/CLEAR] Clear database and start new!
Read start.ans after clear?

○ Read file
○ Do not read file

OK   Cancel   Help

Clears (zeros out) the database stored in memory. Clearing the database has the same effect as leaving and reentering the ANSYS program, but does not require you to exit.

Example0200

Computational Mechanics, AAU, Esbjerg