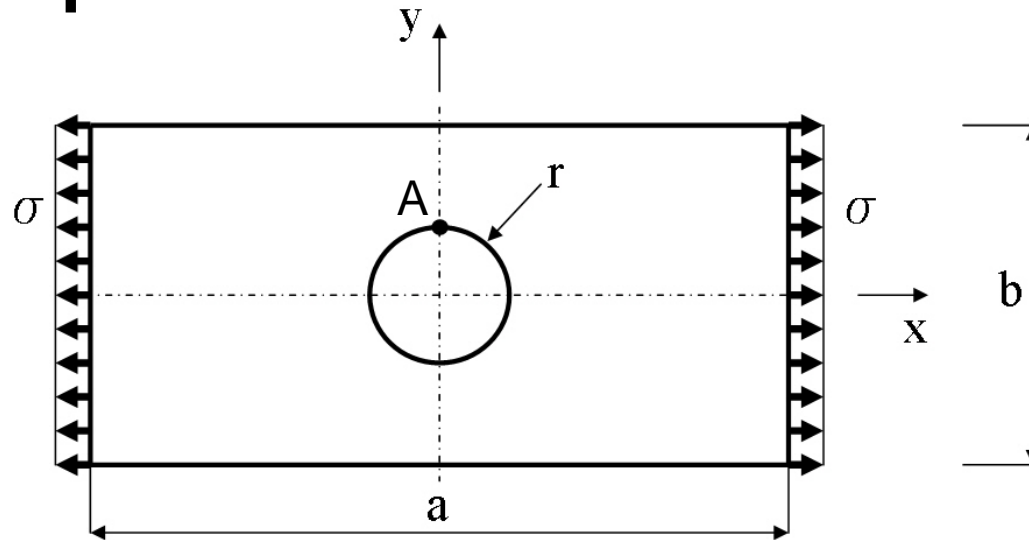


Course in ANSYS

Example0240

Example – Plate with a hole



Objective:

Determine the maximum stress in the x-direction for point A and display the deformation figure

Tasks:

How should this be modelled?

Compare results with results obtained from norm calculations?

Topics:

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

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

$$\nu = 0.3$$

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

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

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

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

$$\sigma = 100 \text{ N/mm}^2$$

Example - title

Utility Menu > File > Change Jobname

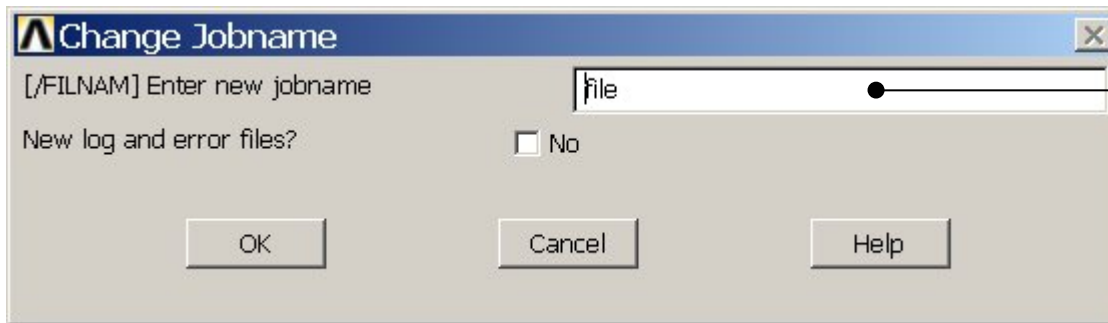


GUI

/jobname, Example0240



Command line entry

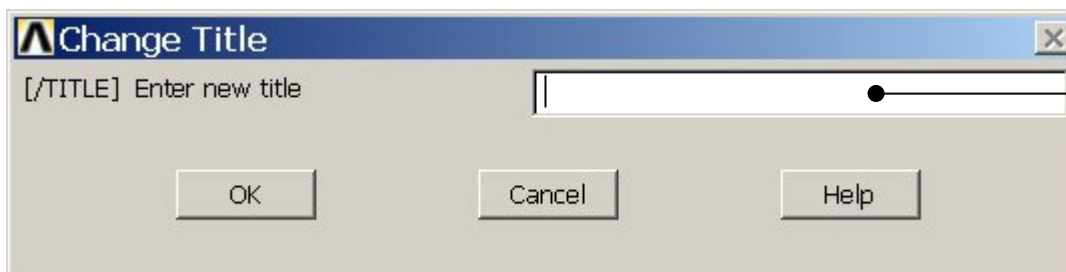


Enter: Example0240

Utility Menu > File > Change Title

/title, Plate with a hole

Enter: Plate with a hole



Example – Areas Rectangle

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

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

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 fields. Arrows point from text labels to these fields: 'Enter 0 or leave empty' points to the first field of the X-coordinates row, 'Enter 100' points to the second field of the X-coordinates row, 'Enter 0 or leave empty' points to the first field of the Y-coordinates row, and 'Enter 50' points to the second field 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. On the left, the ANSYS Main Menu is visible, with the path 'Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions' highlighted. 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 'Example0240' and 'Computational mechanics, AAU, Esbjerg' are visible, along with the page number '4'.

Enter 0 or leave empty

Enter 100

Enter 0 or leave empty

Enter 50

Press OK

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

Example0240

Computational mechanics, AAU, Esbjerg

4

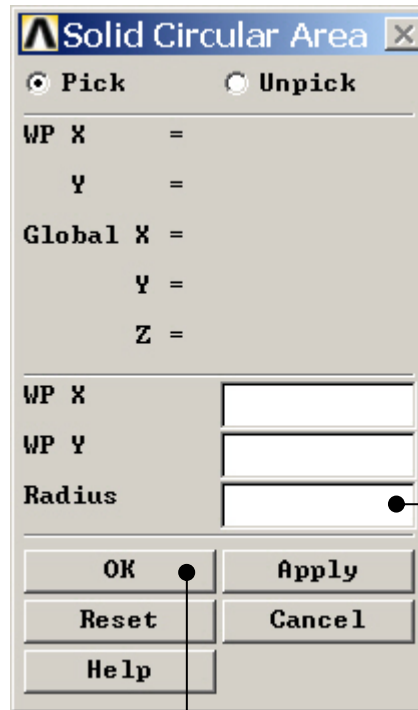
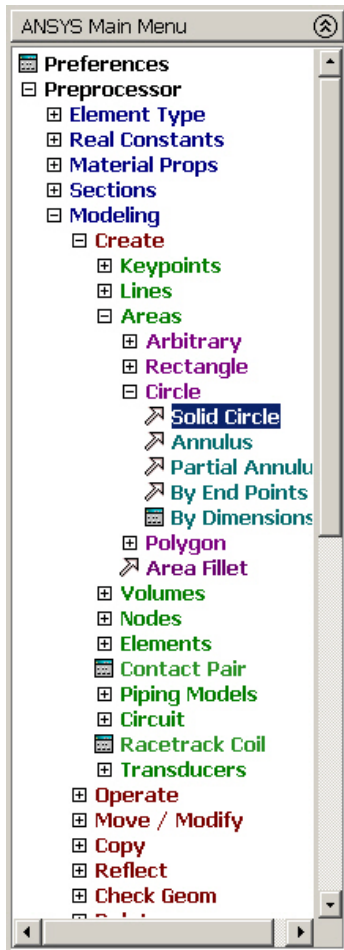
Example – Areas Rectangle



Example – Areas Circle

Preprocessor > Modeling > Create > Areas > Circle > Solid Circle

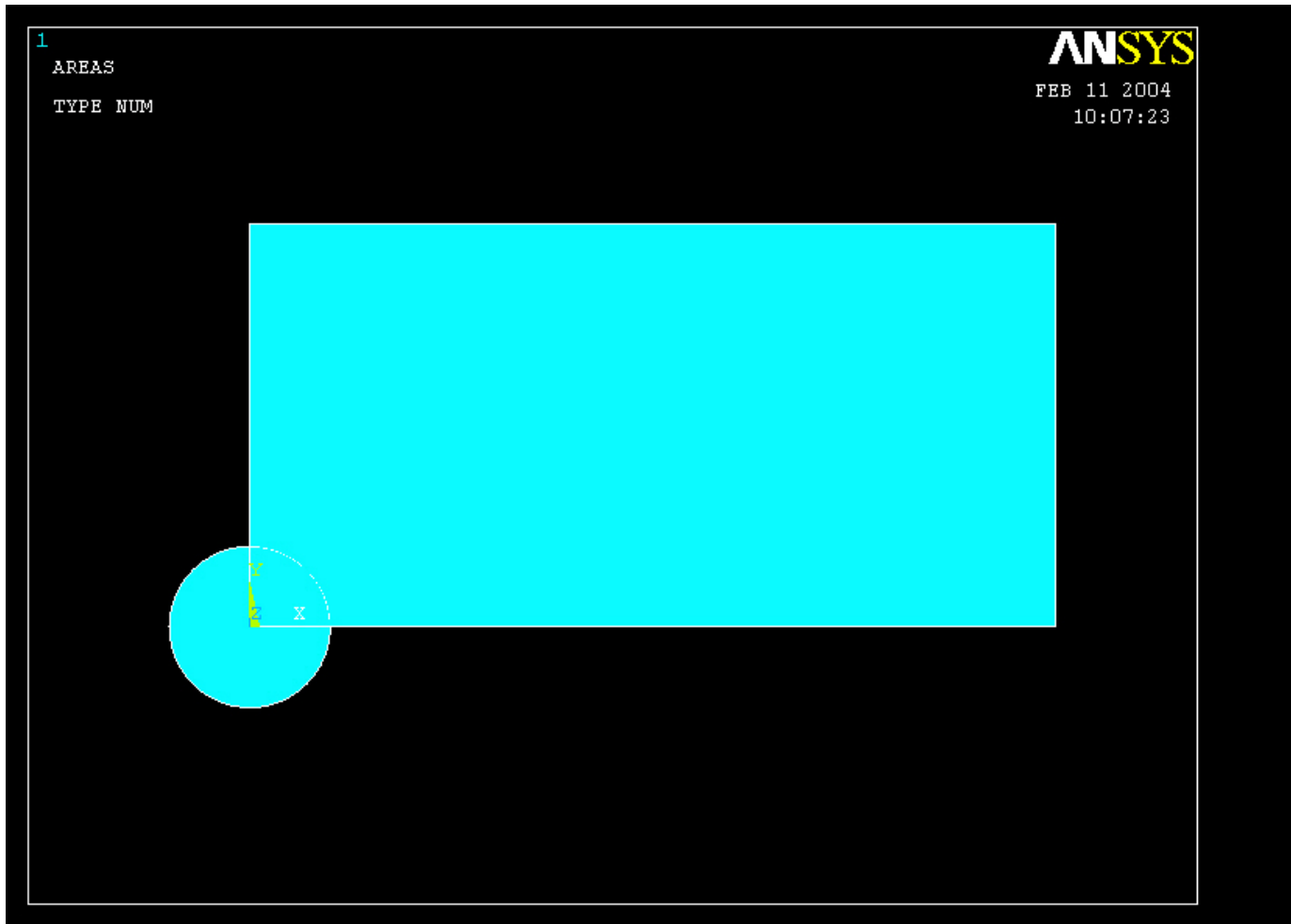
Create an area given by $(X,Y)=(0, 0)$ and Radius=10



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

Example0240

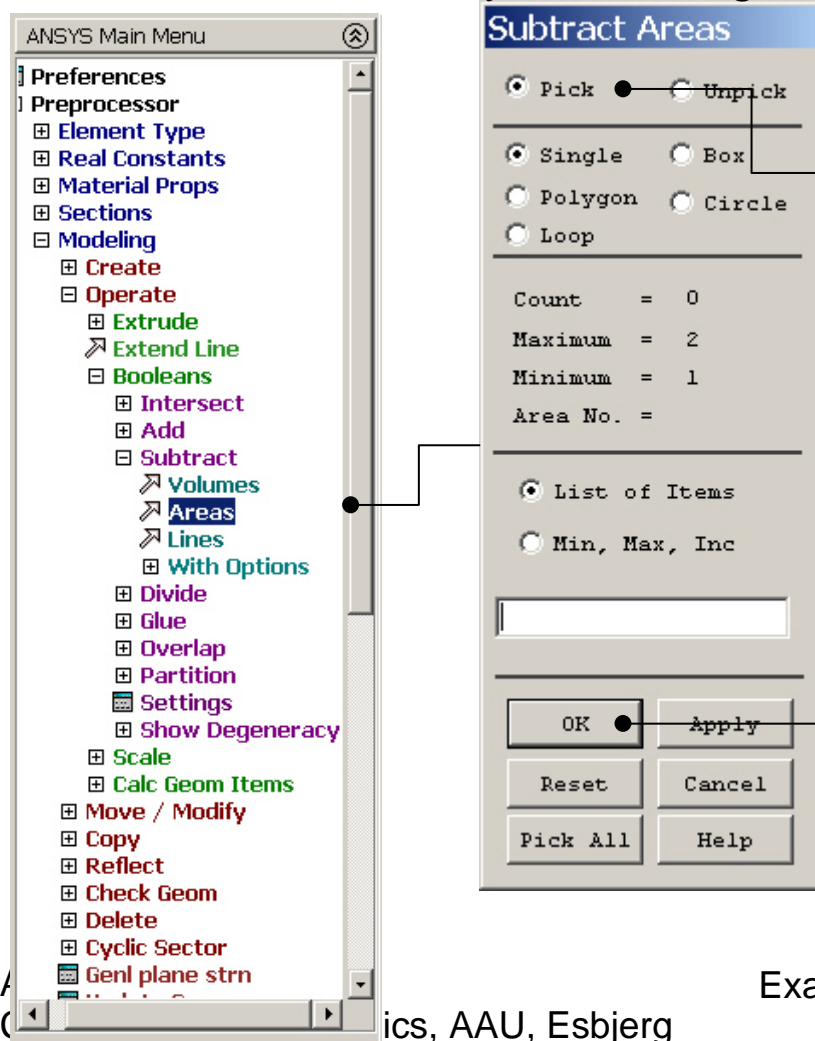
Example - Area



Example - Operate

Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

Create the final area by subtracting the circular area from the rectangular area



Note: Bottom left corner of ANSYS GUI

[ASBA] Pick or enter base areas from which to subtract

Select the rectangular area and press OK

Note: Bottom left corner of ANSYS GUI

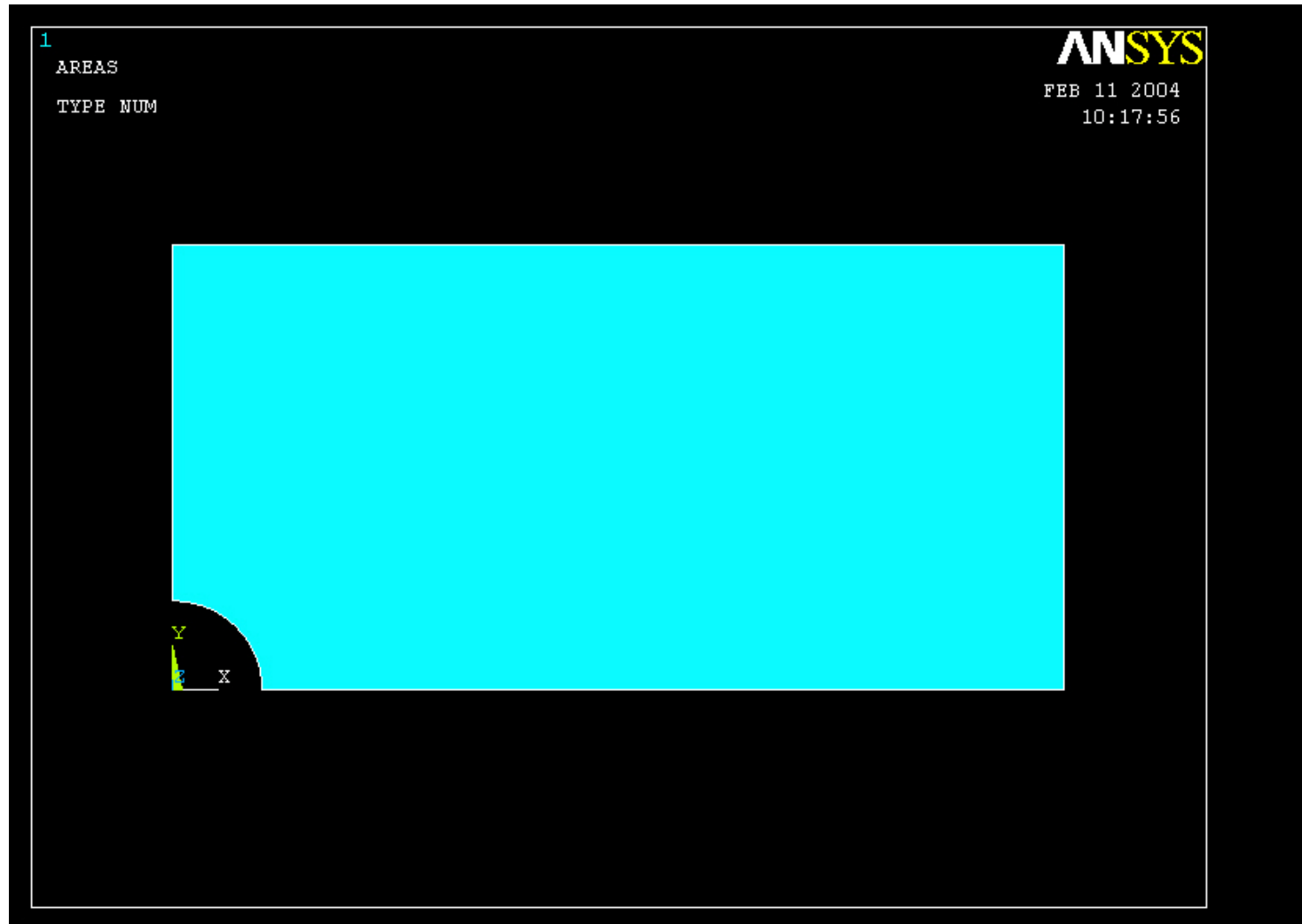
Pick or enter areas to be subtracted

Select the circular area

Press OK

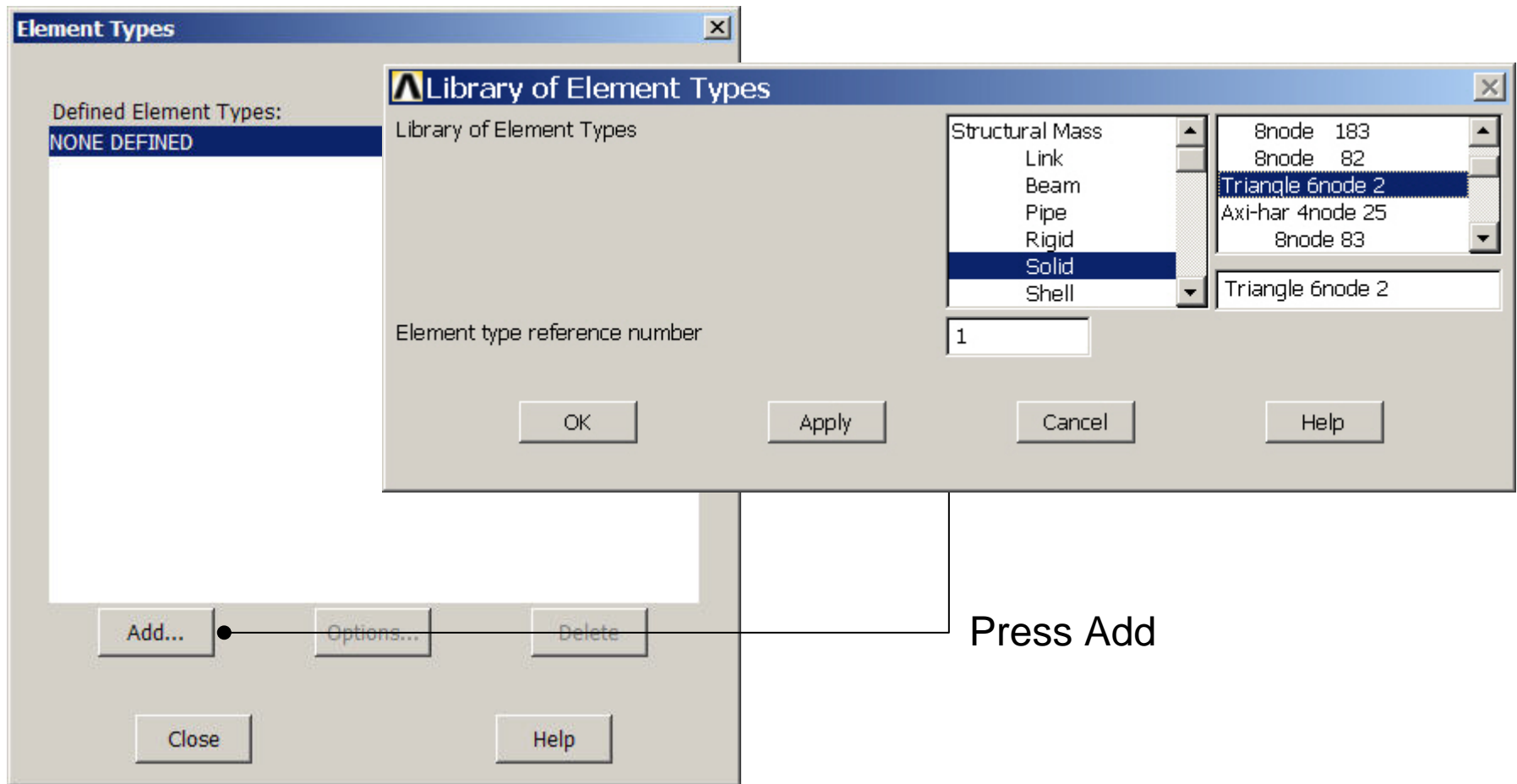
Example0240

Example – Areas



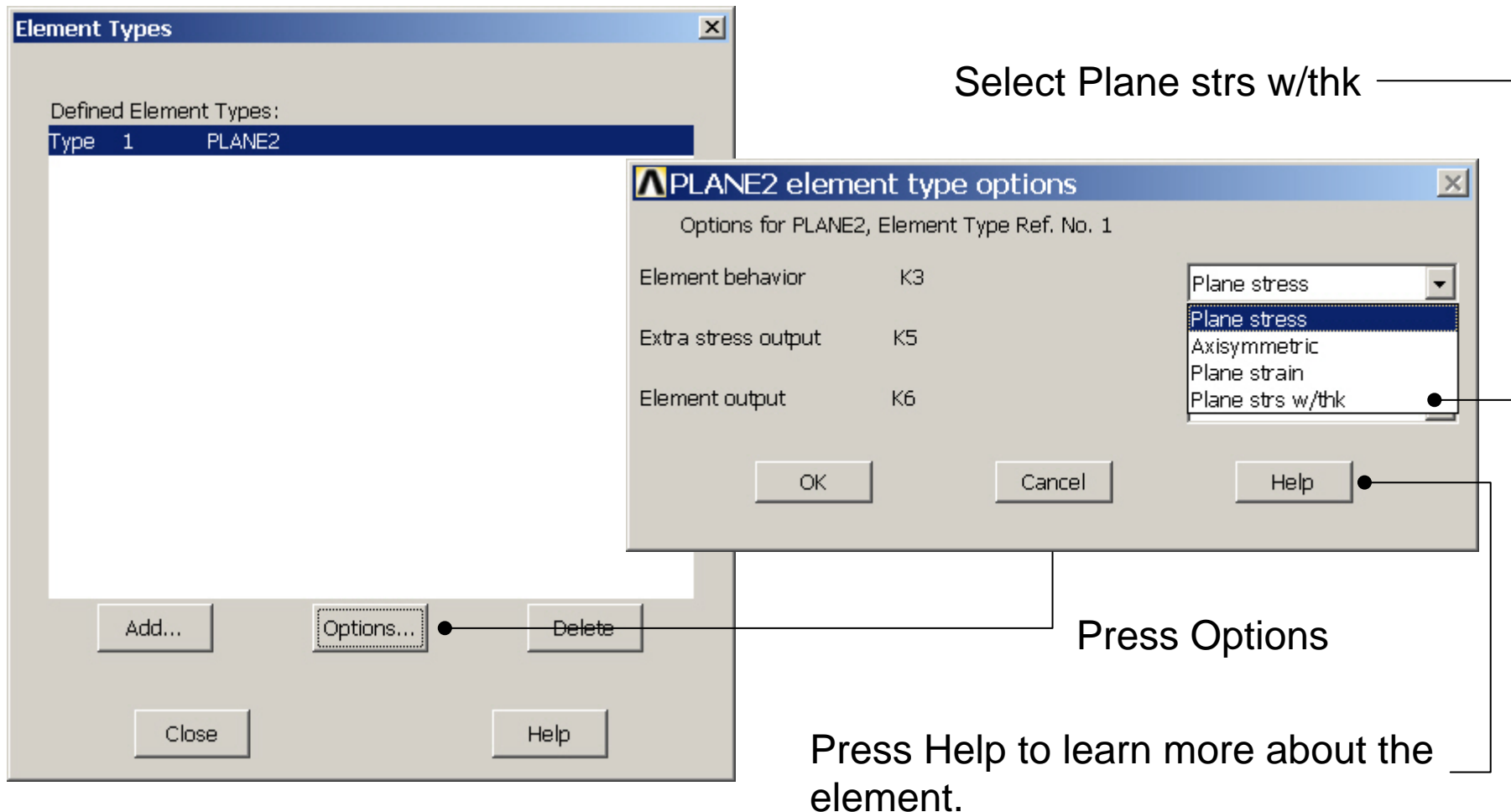
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



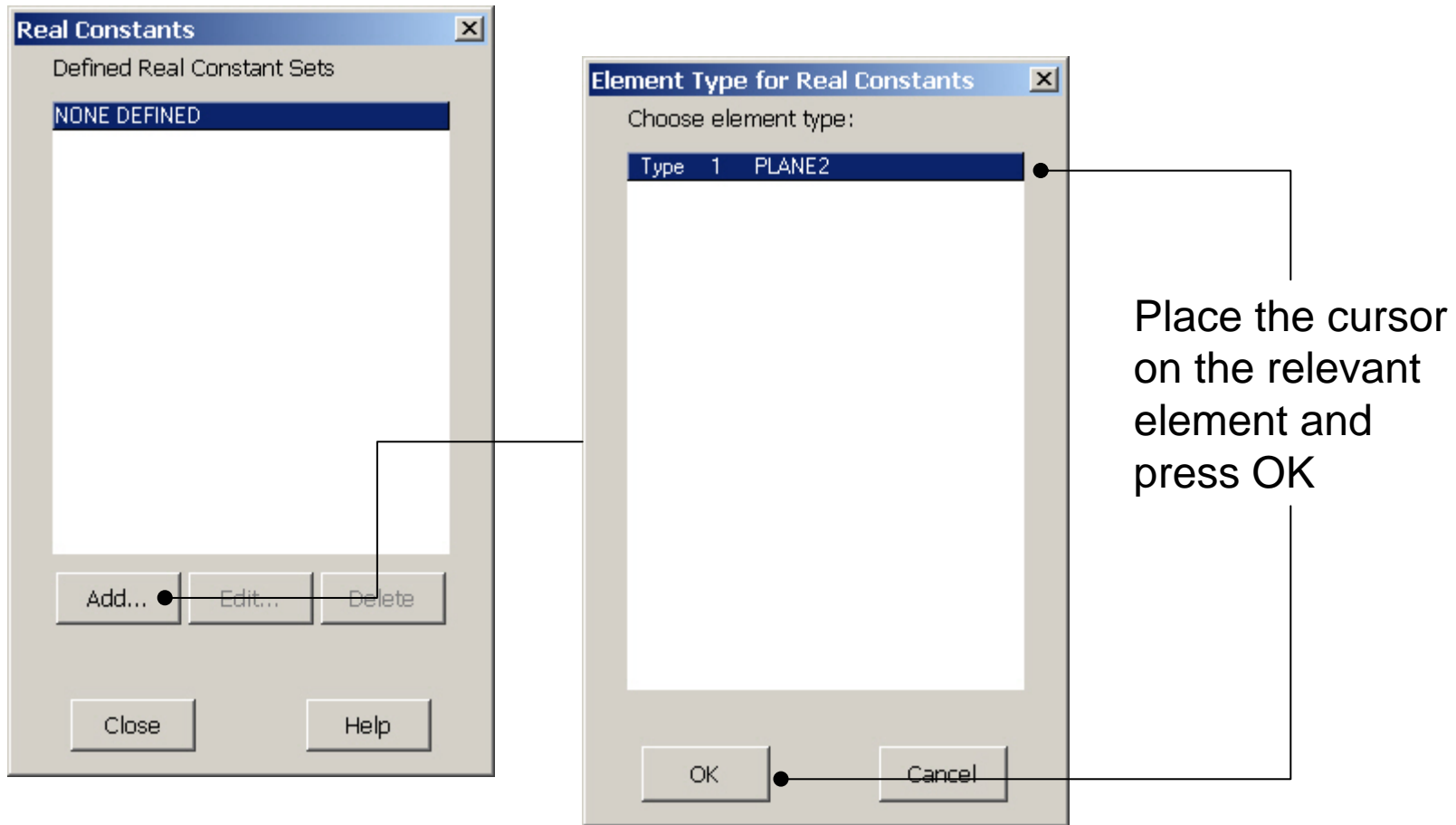
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



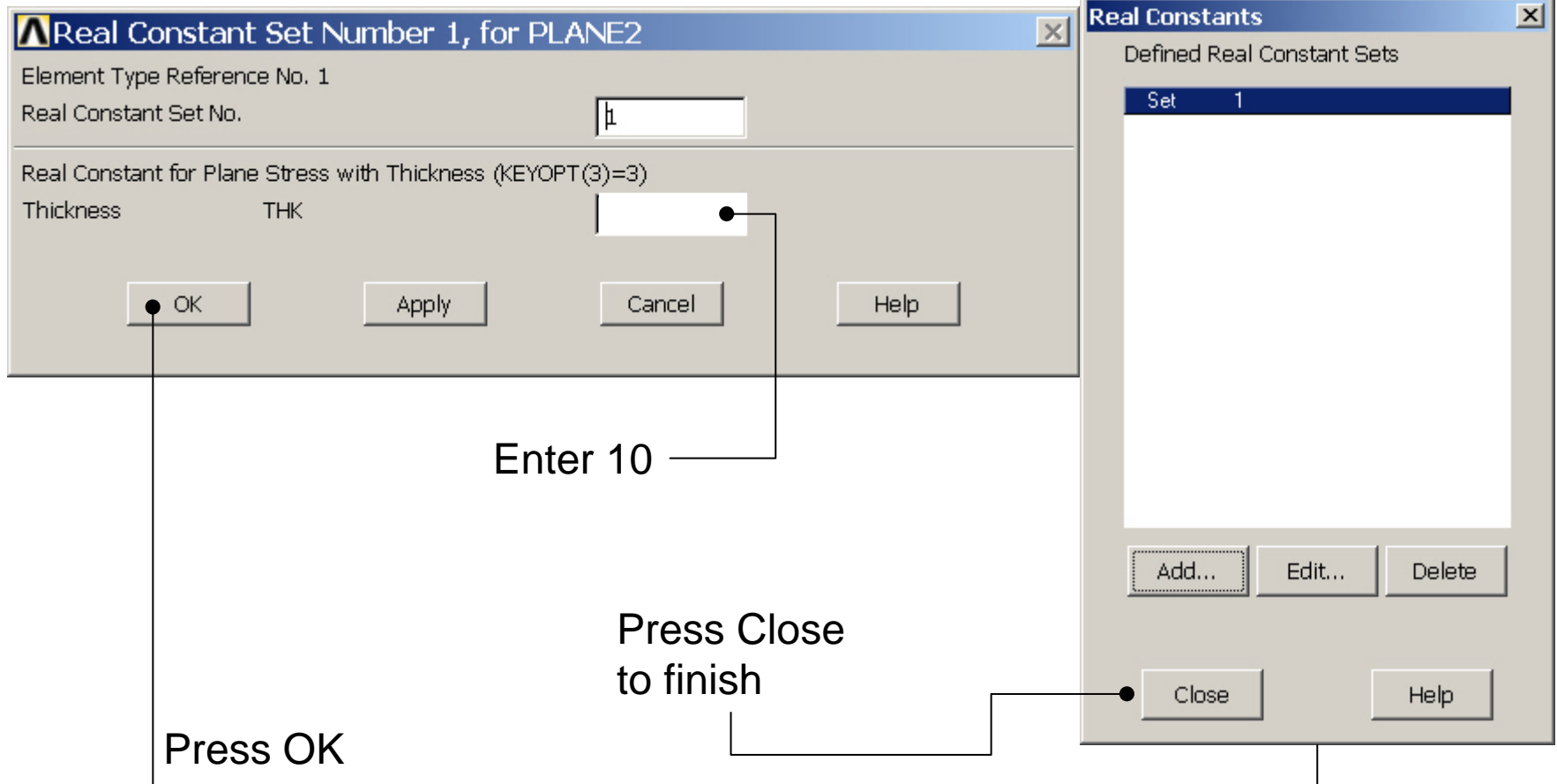
Example – Real Constants

Preprocessor > Real Constants > Add



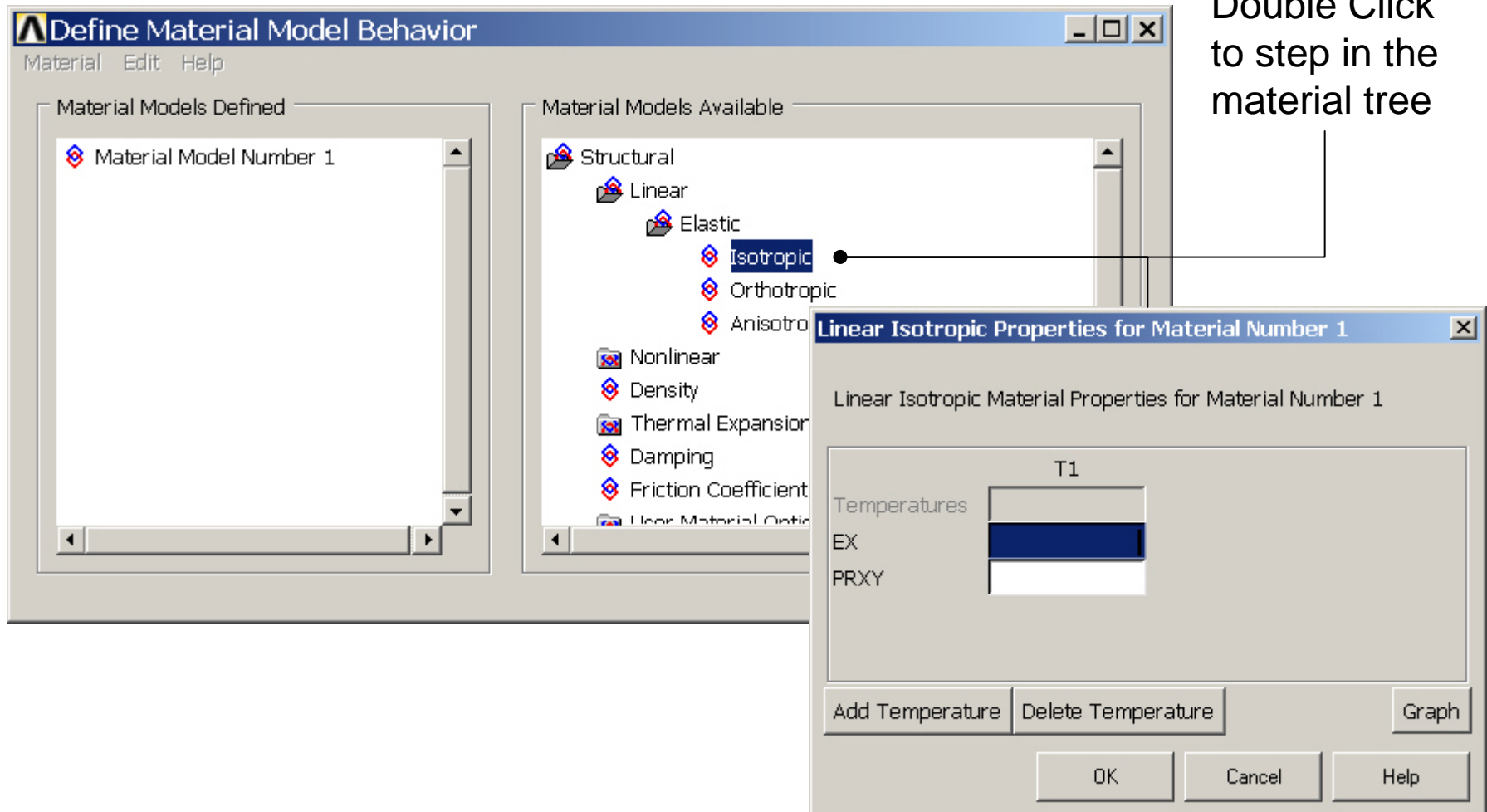
Example - Real Constants

Preprocessor > Real Constants > Add



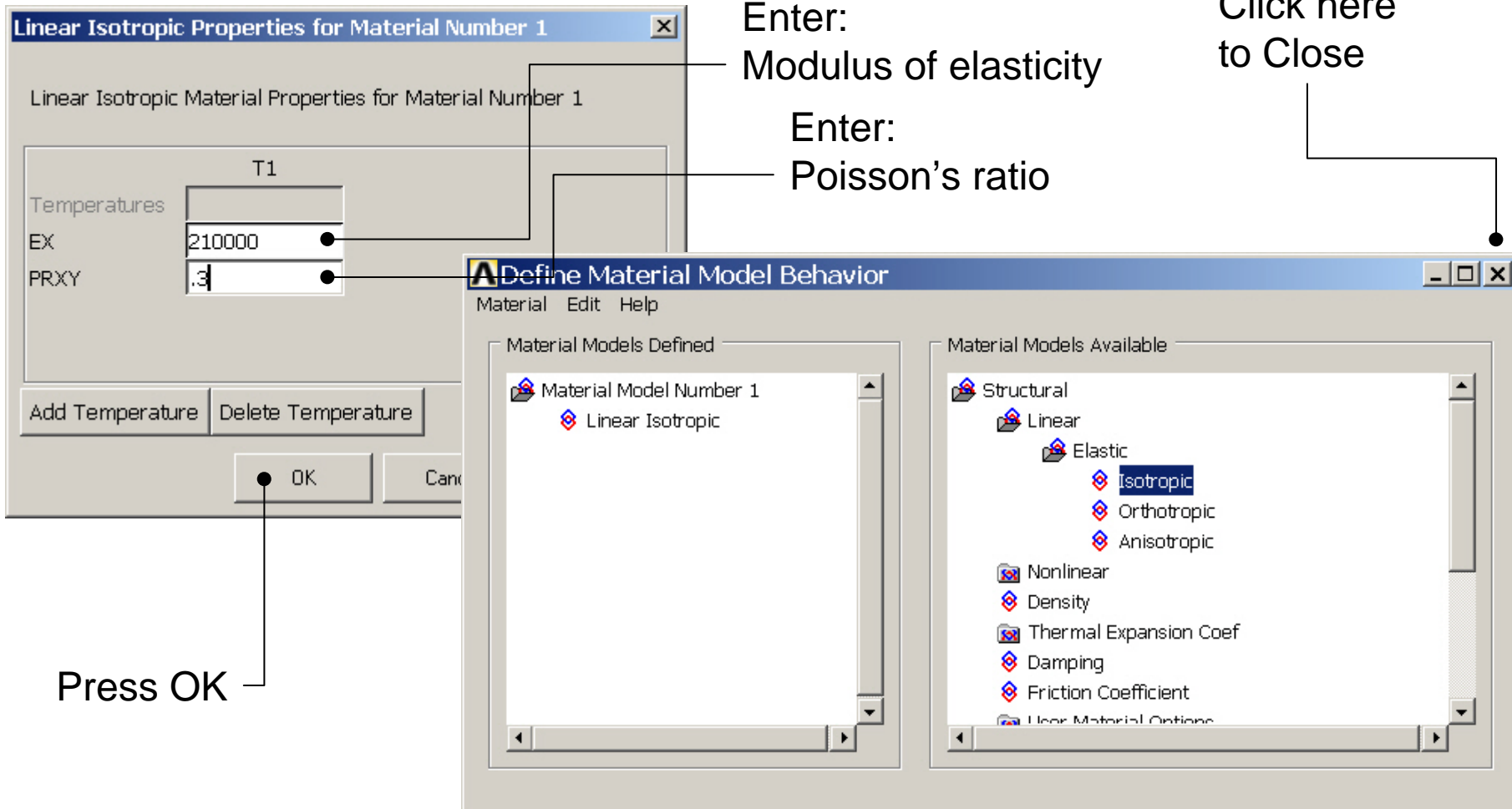
Example - Material Properties

Preprocessor > Material Props > Material Models



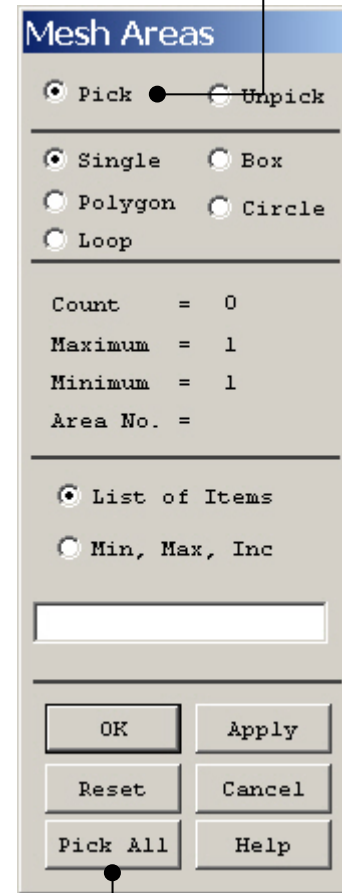
Example - Material Properties

Preprocessor > Material Props > Material Models



Example – Free Meshing

Preprocessor > Meshing > Mesh > Areas > Free

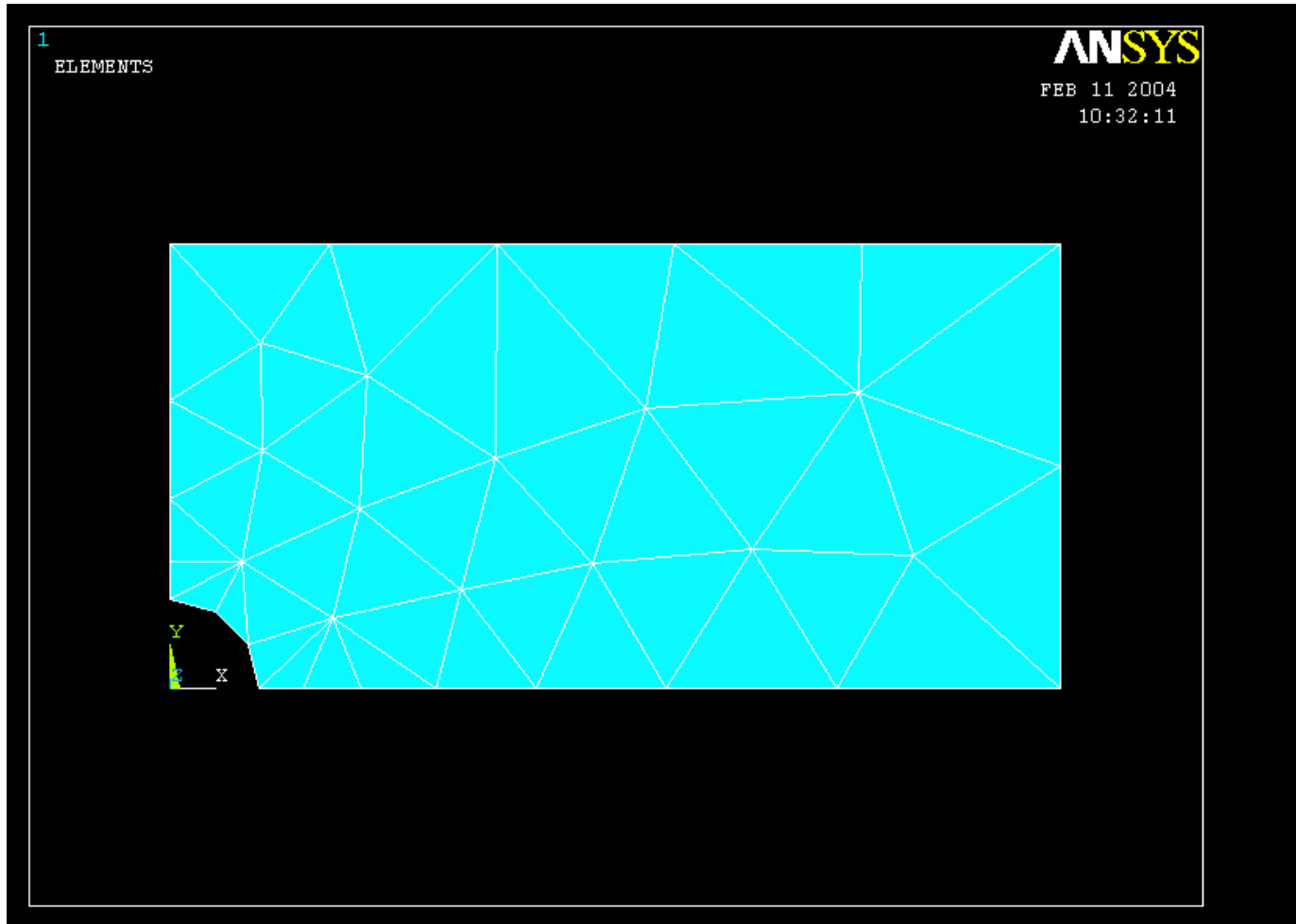


Select individual areas to be meshed

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

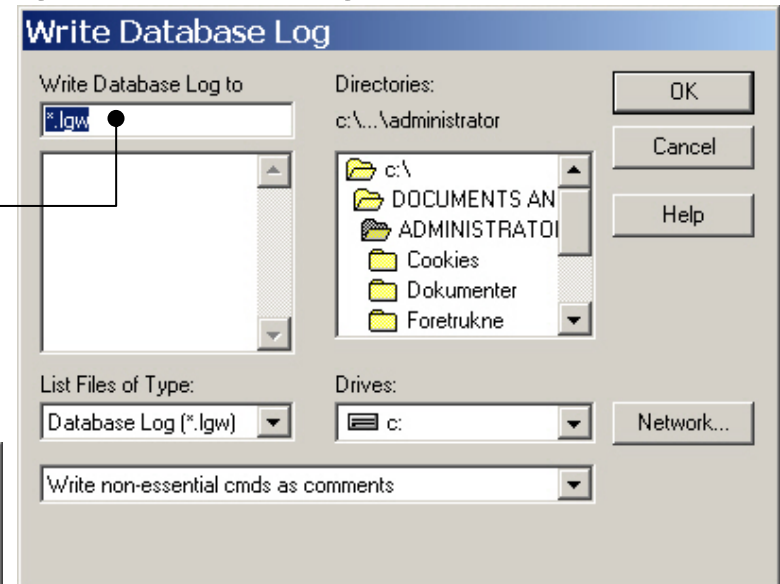
Example – Free Mesh



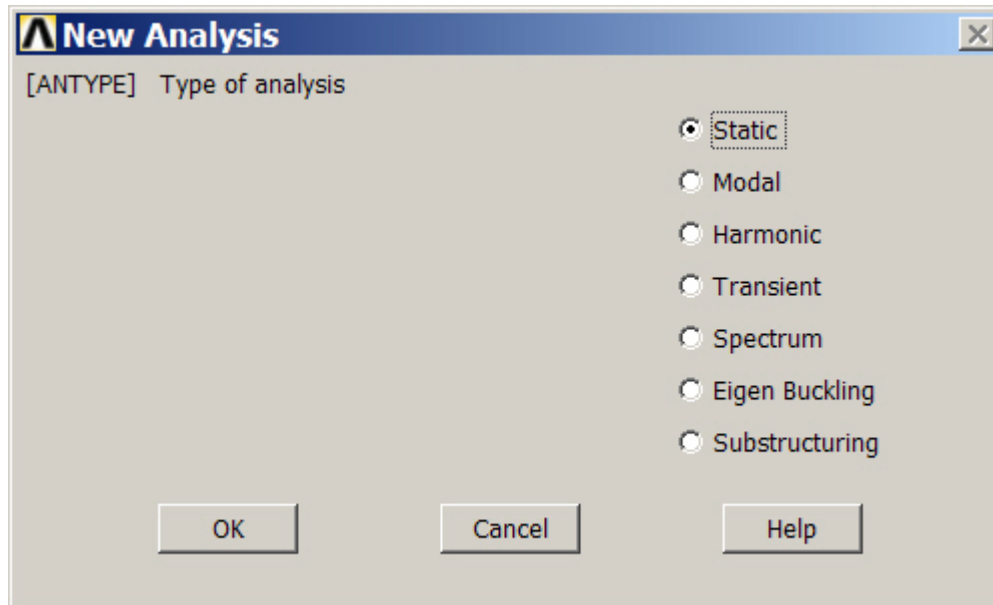
Example – Analysis Type

File > Write DB log file

Enter “example0240.lgw”

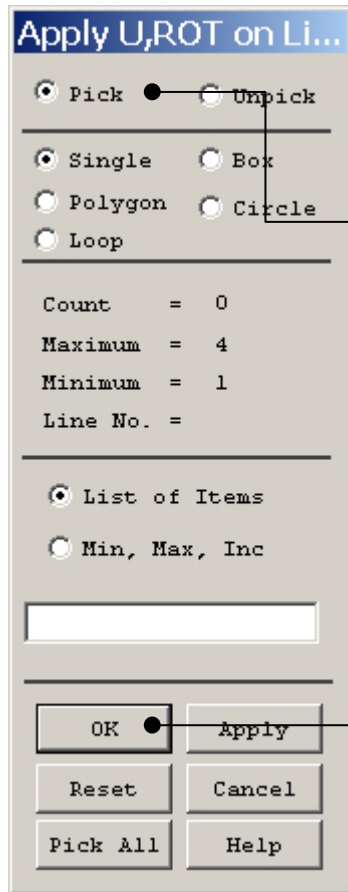


Solution > Analysis Type > New Analysis



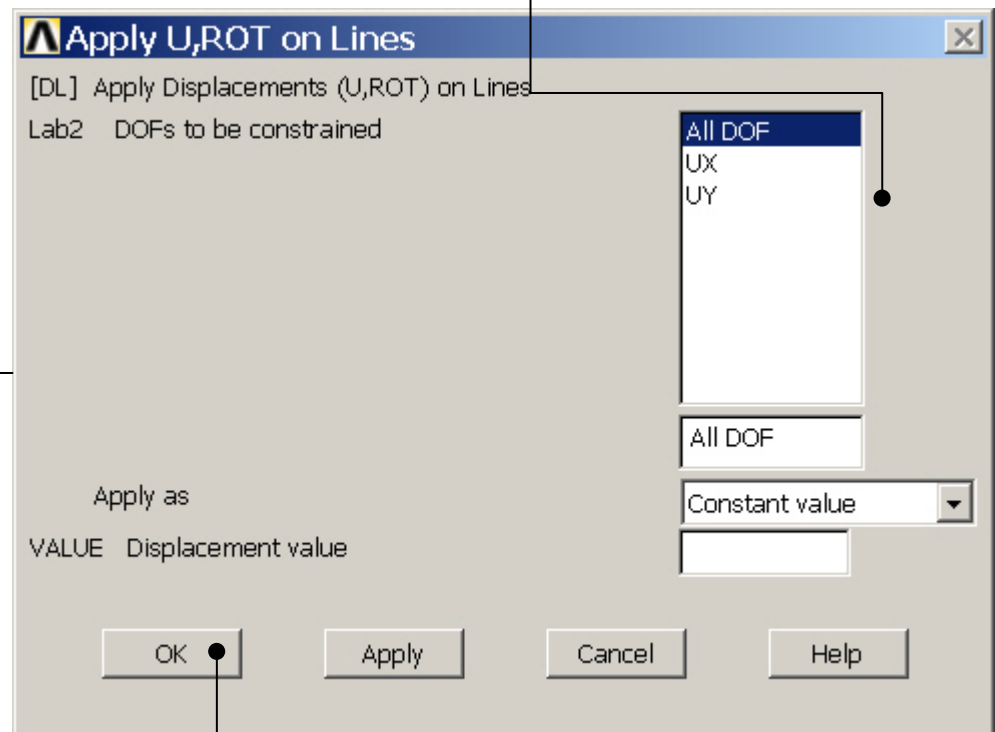
Example – Define Loads

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



Select the
bottom straight
line

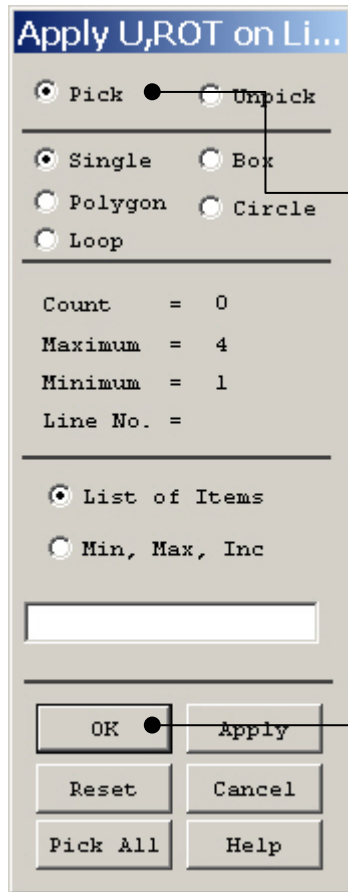
Select UY to fix the beam in the y-direction



Press OK

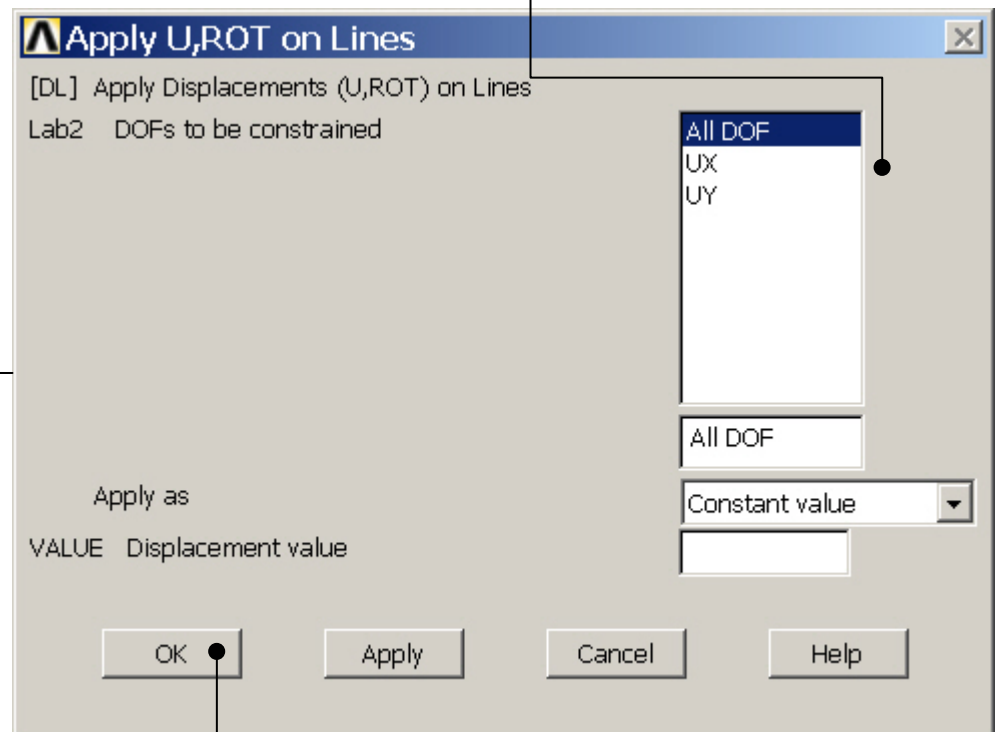
Example – Define Loads

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



Select the left straight line

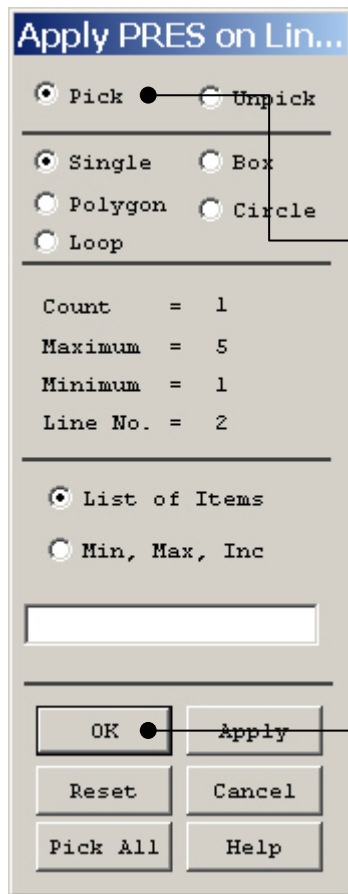
Select UX to fix the beam in the x-direction



Press OK

Example – Define Loads

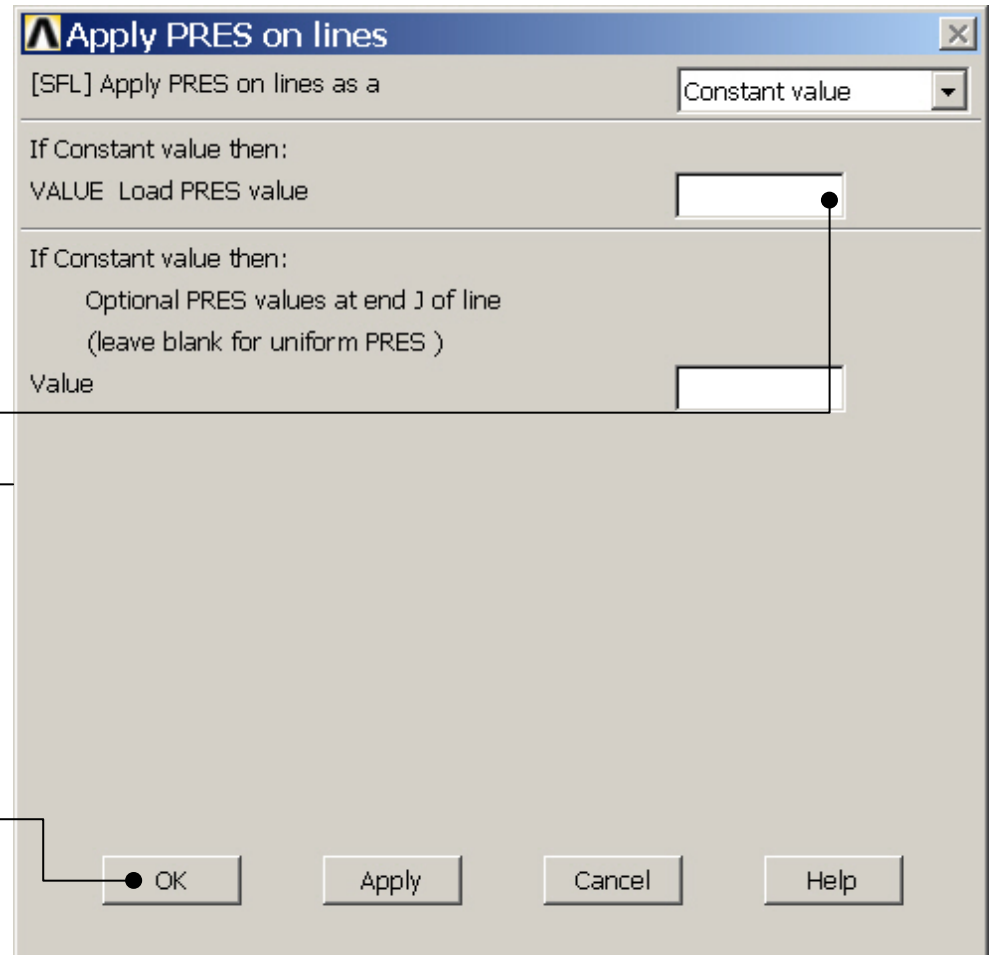
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the
right
straight line

Enter -100

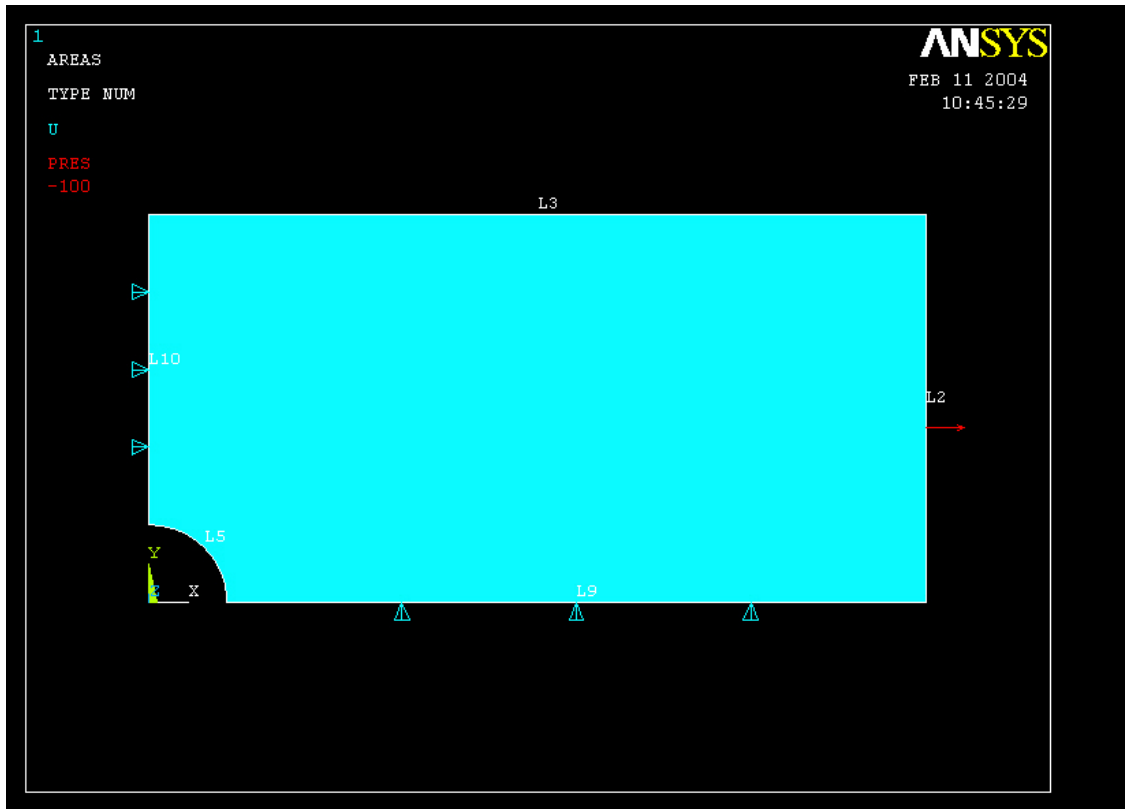
Press OK
to finish



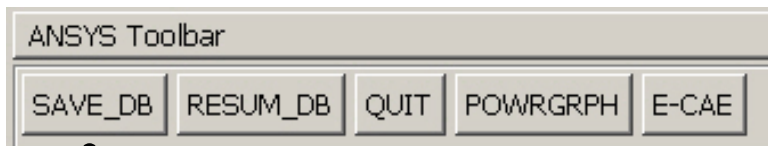
Example0240

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

Example - Save



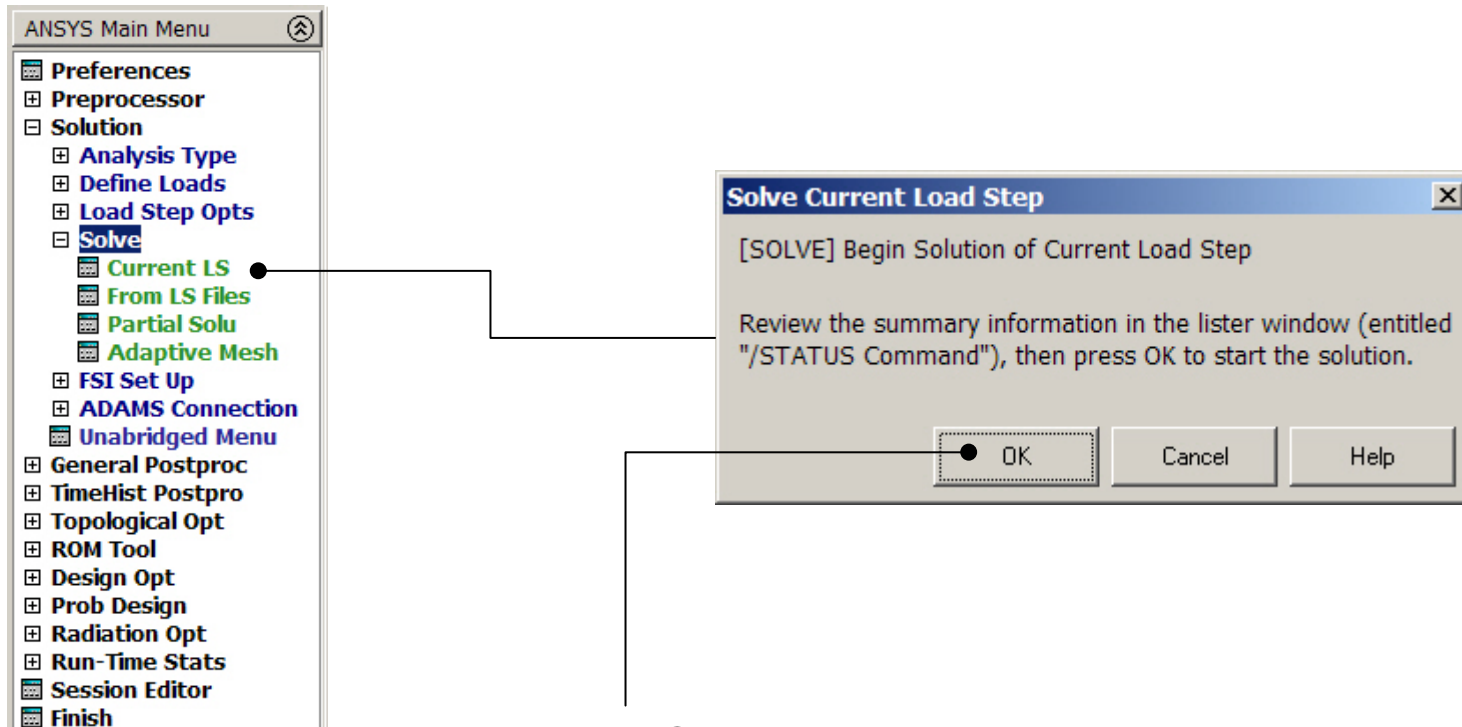
Display of Analysis model



Save the model

Example - Solve

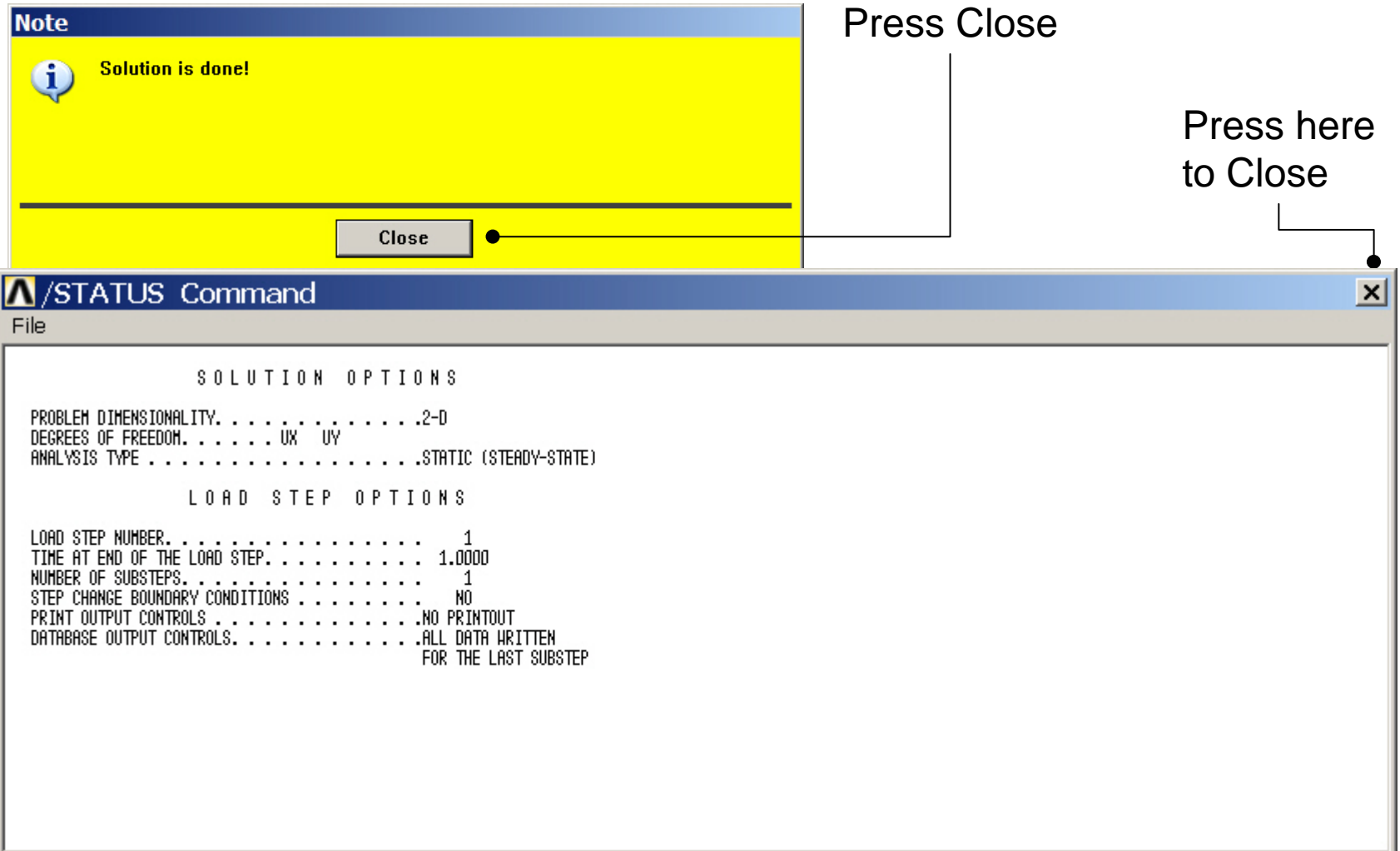
Solution > Solve > Current LS



Press OK

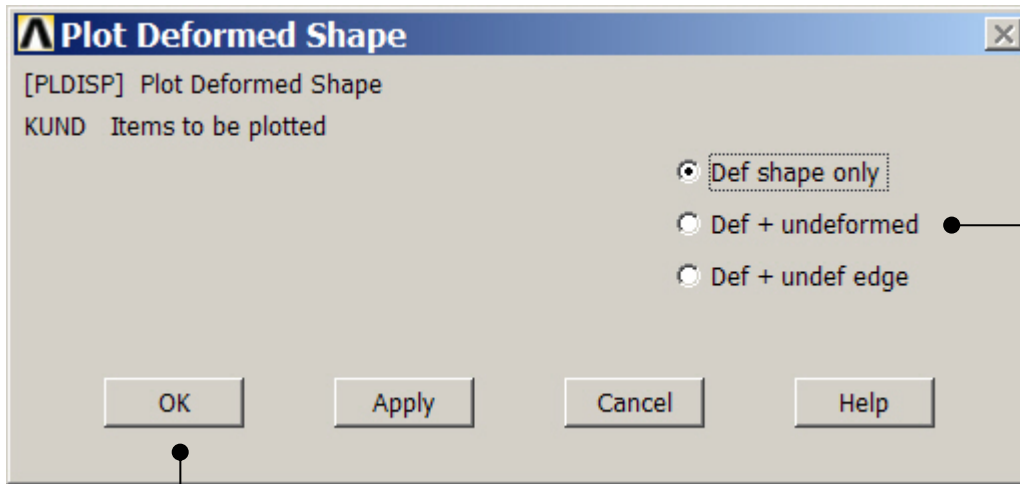
Example0240

Example - Solve



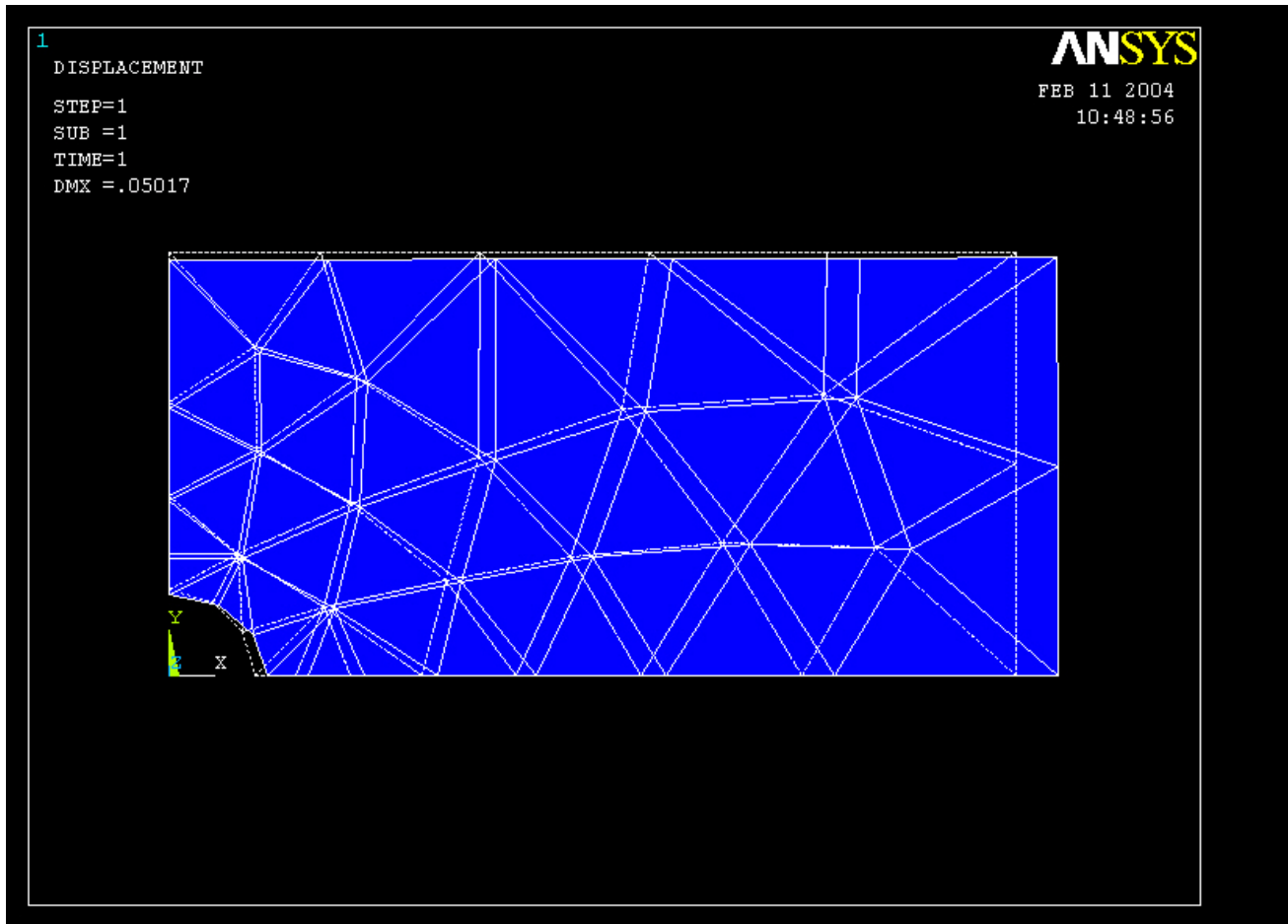
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Select "Def+undeformed"
and Press OK

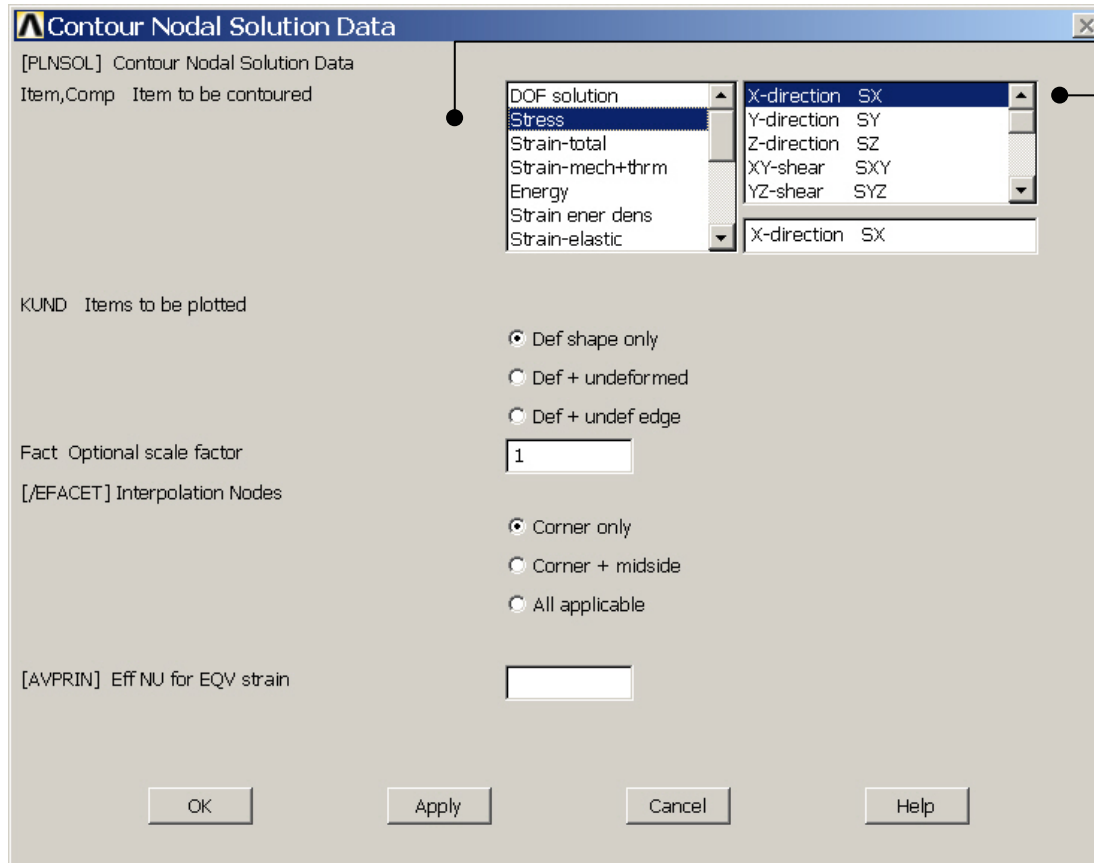
Example - PostProcessing



Read Maximum displacement: DMX

Example – Contour Plot

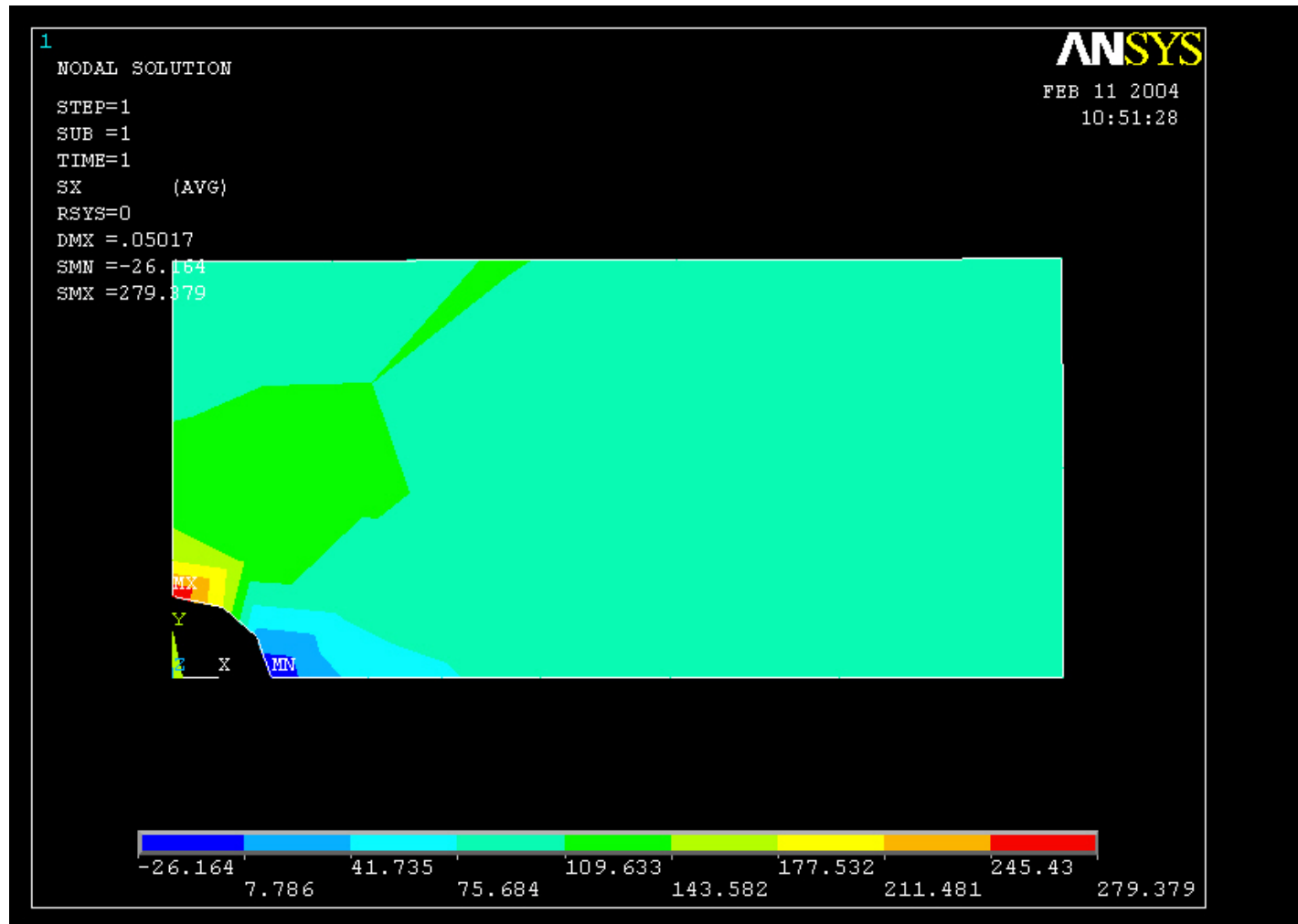
General Postproc > Plot Results > Contour Plot > Nodal Sol



Select Stress

Select SX for stresses in x-direction

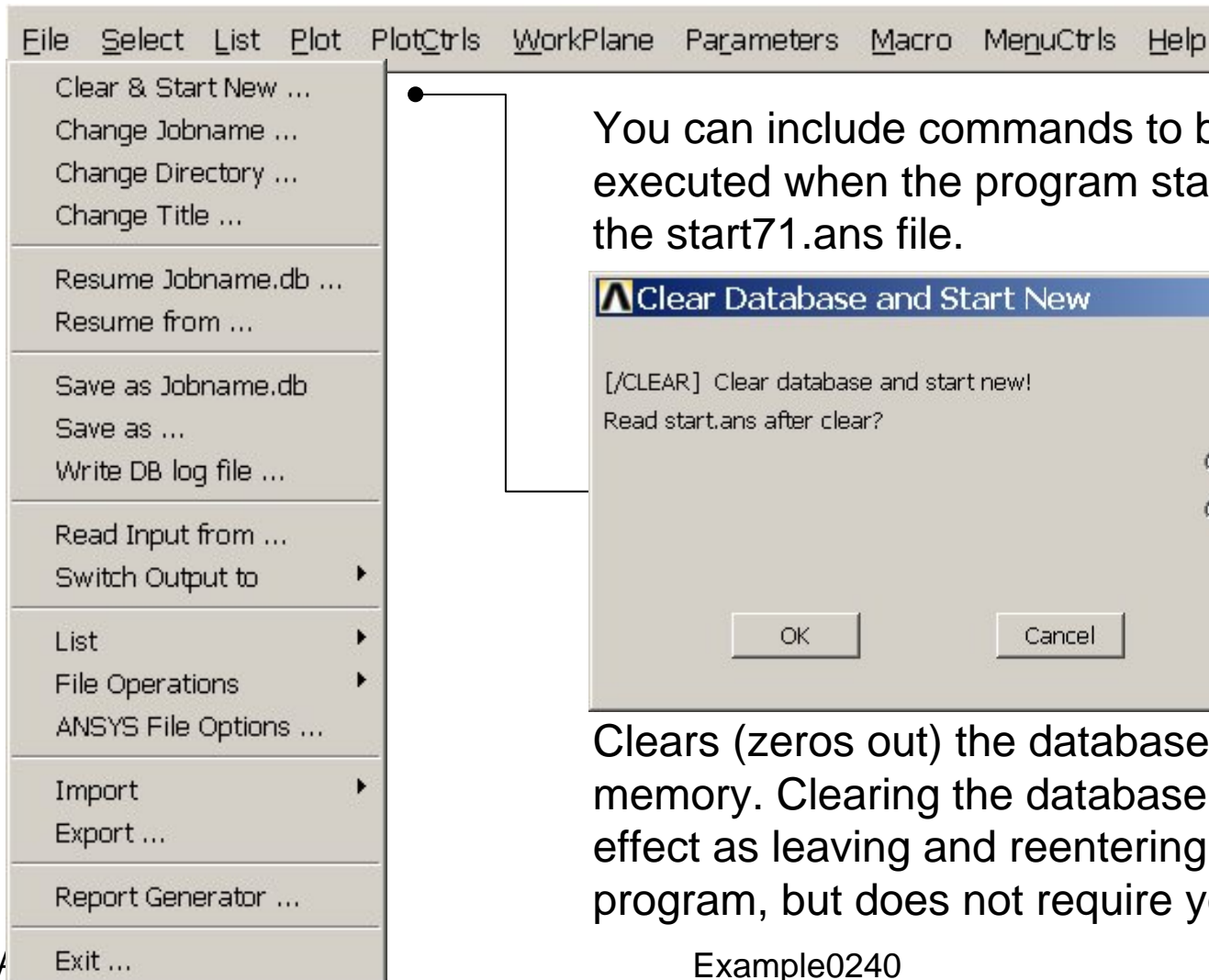
Example – Contour Plot



Example – Comments/Questions

- Change the element size and rerun?
- The “example0240.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



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

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.