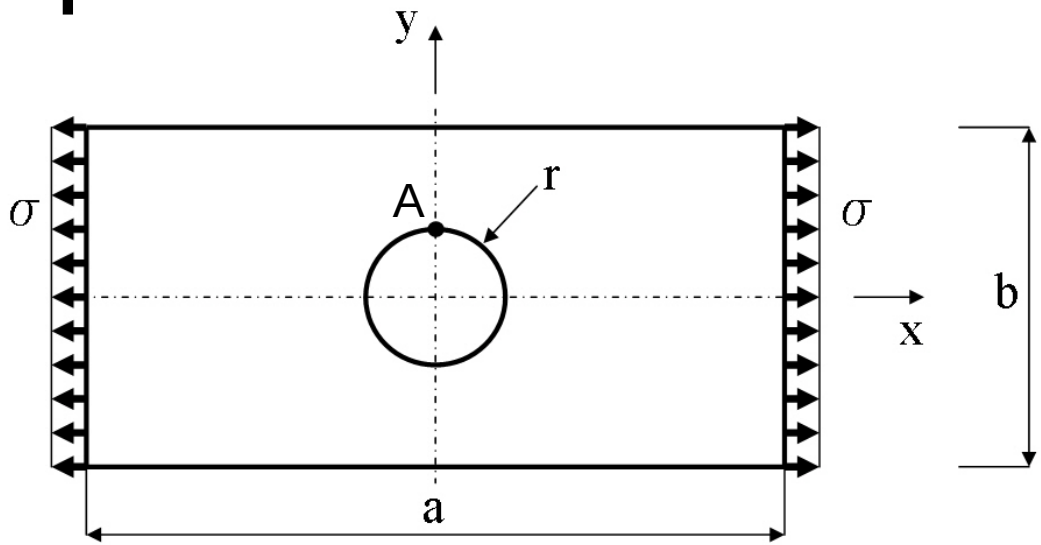
Course in ANSYS

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

$E = 210000 N/mm^2$

$\nu = 0.3$

a = 200mm

b = 100mm

t = 10mm

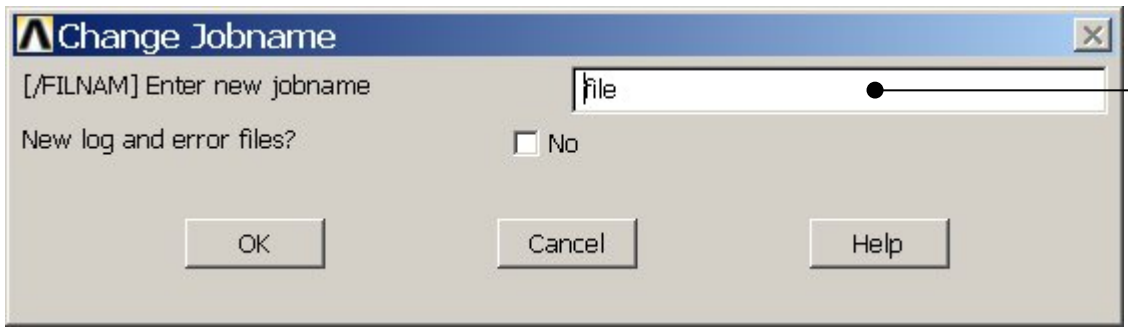
r = 10mm

$\sigma = 100 N/mm^2$

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0240 — Command line entry

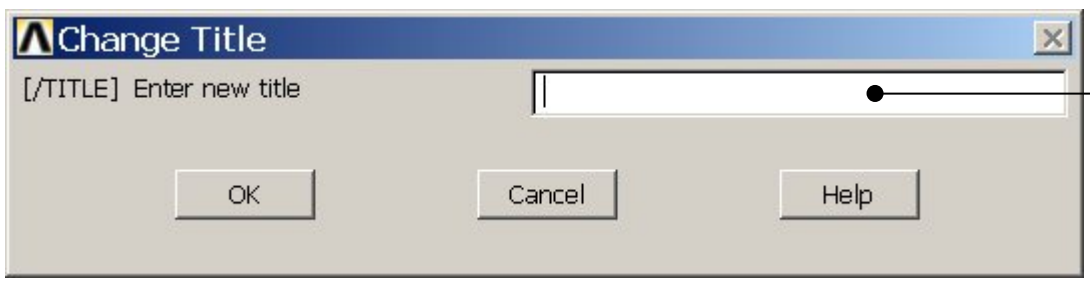


Enter: Example0240

Utility Menu > File > Change Title

/title, Plate with a hole

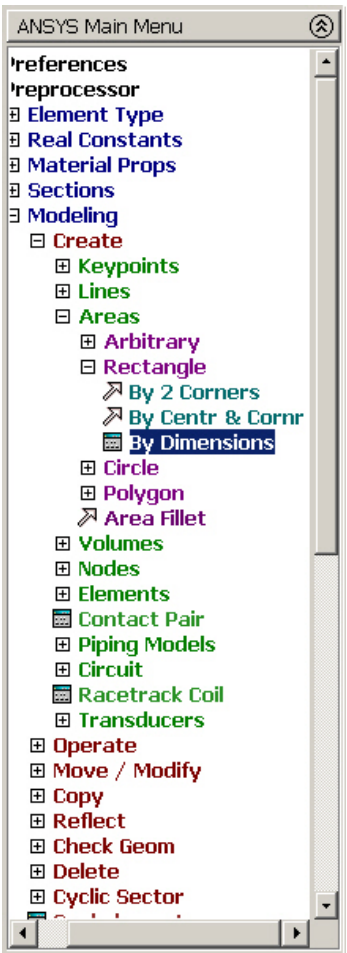
Enter: Plate with a hole

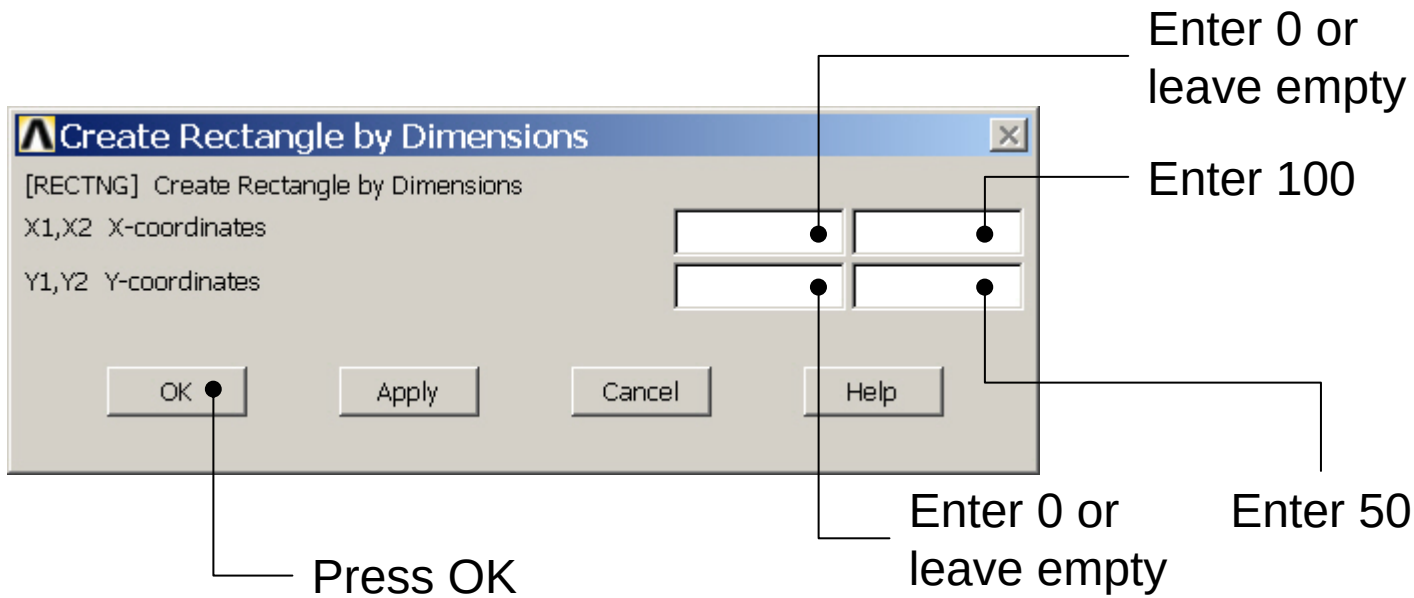


Example – Areas Rectangle

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

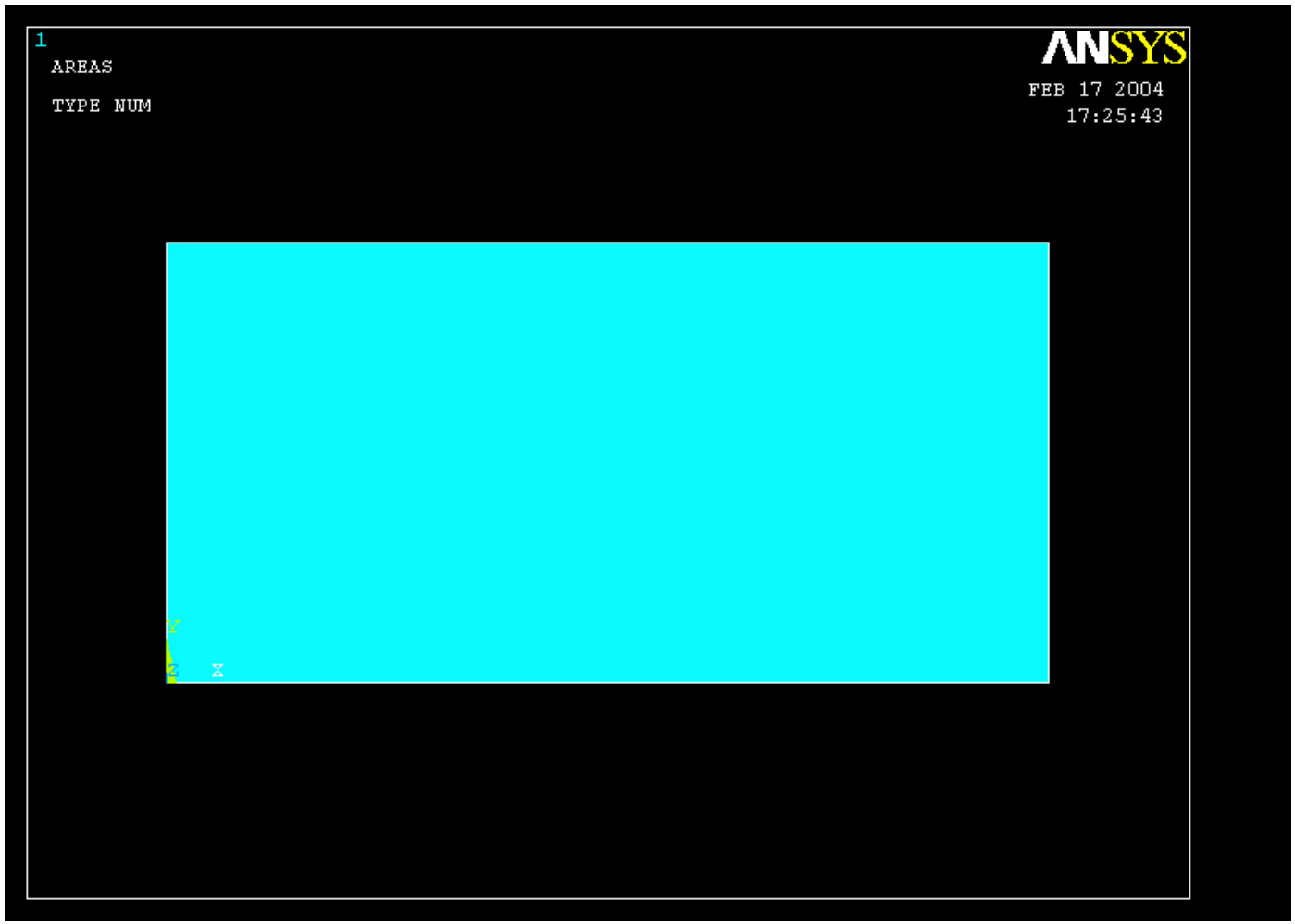
Create an area given by X=(0,100) and Y=(0,50)





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

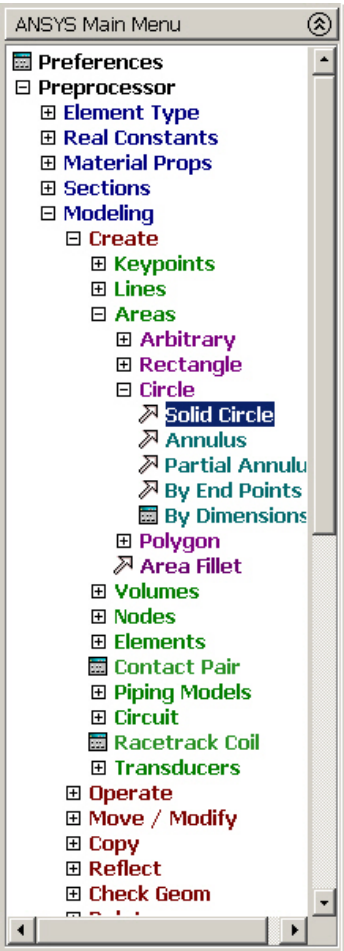
Example – Areas Rectangle

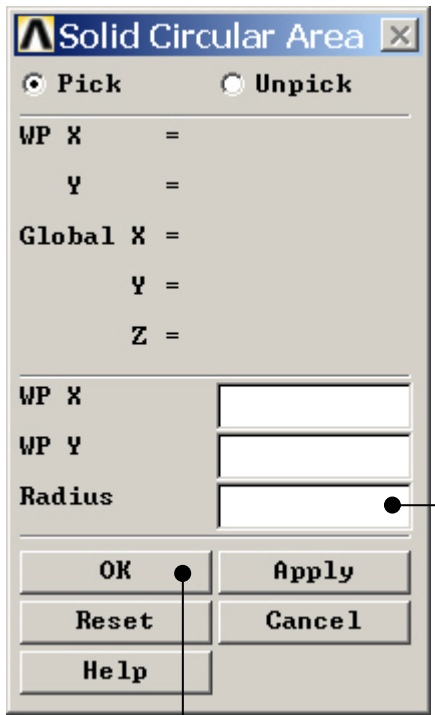


Example – Areas Circle

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

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





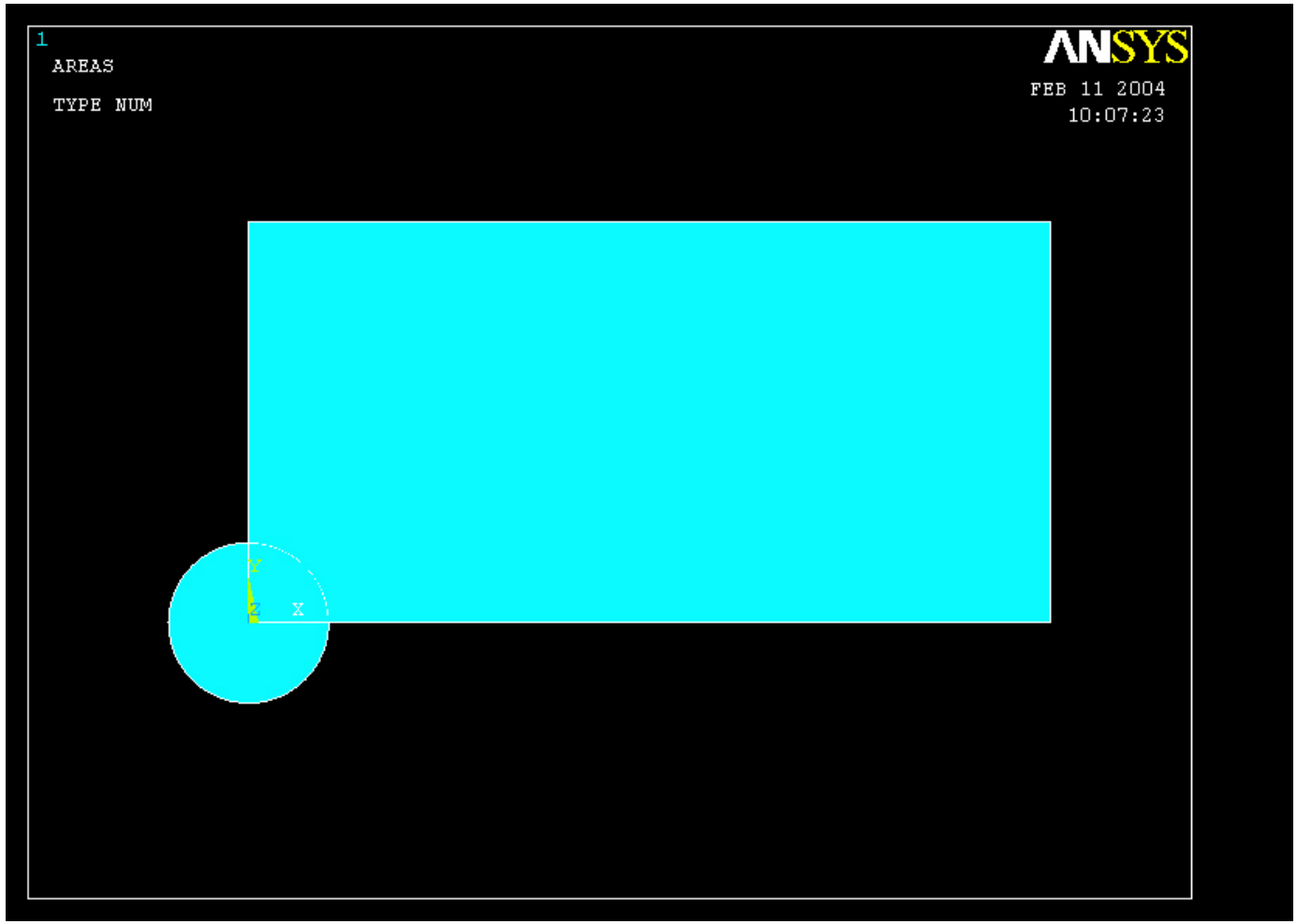
Enter 10

Press OK

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

Example0240

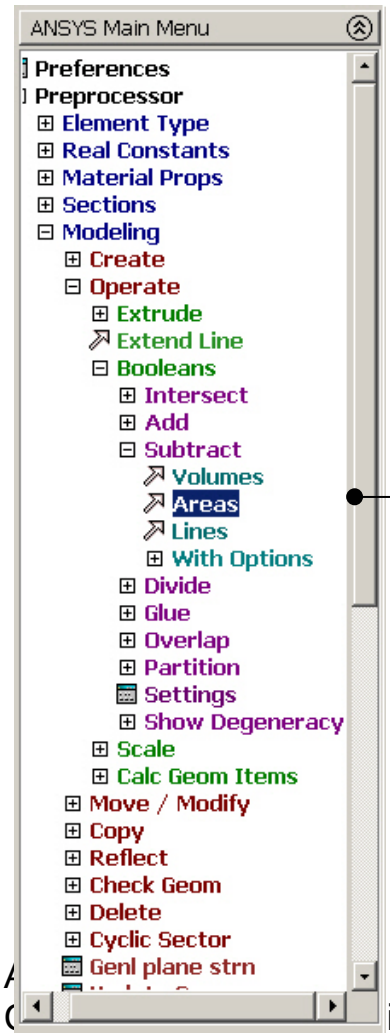
Example - Area

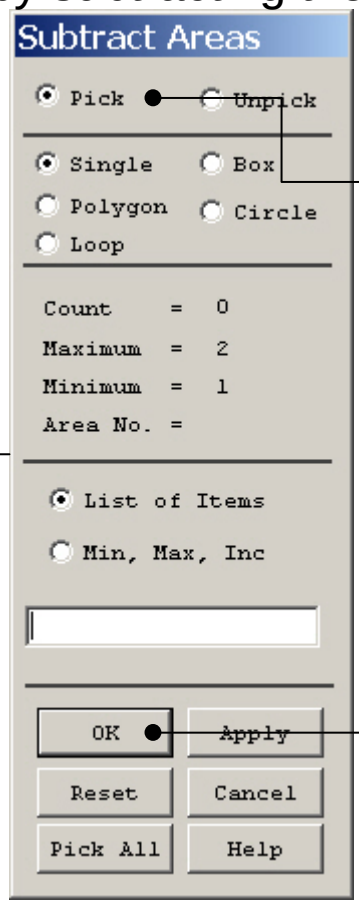


Example - Operate

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

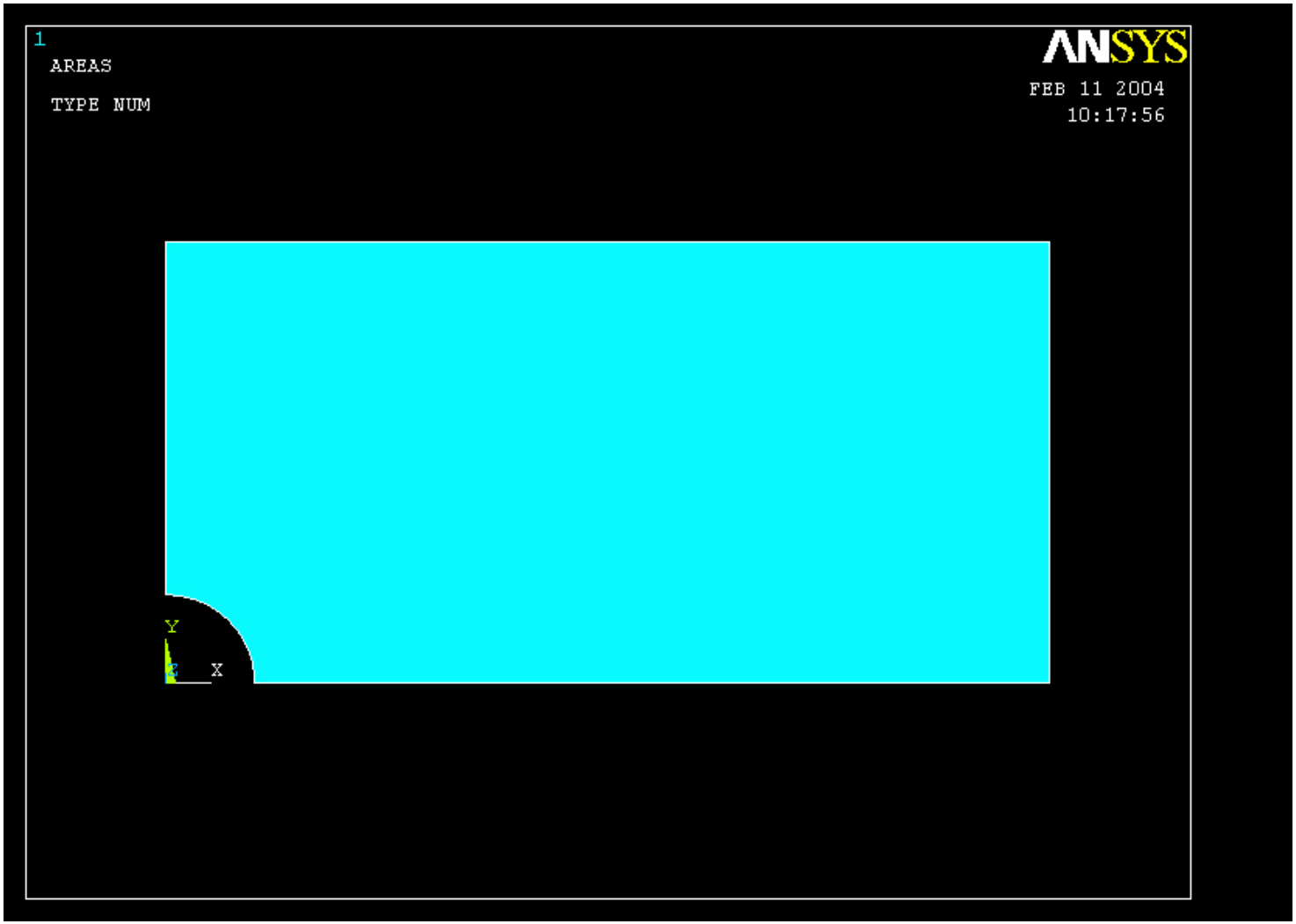
Note: Bottom left corner of ANSYS GUI

Pick or enter areas to be subtracted

Select the circular area

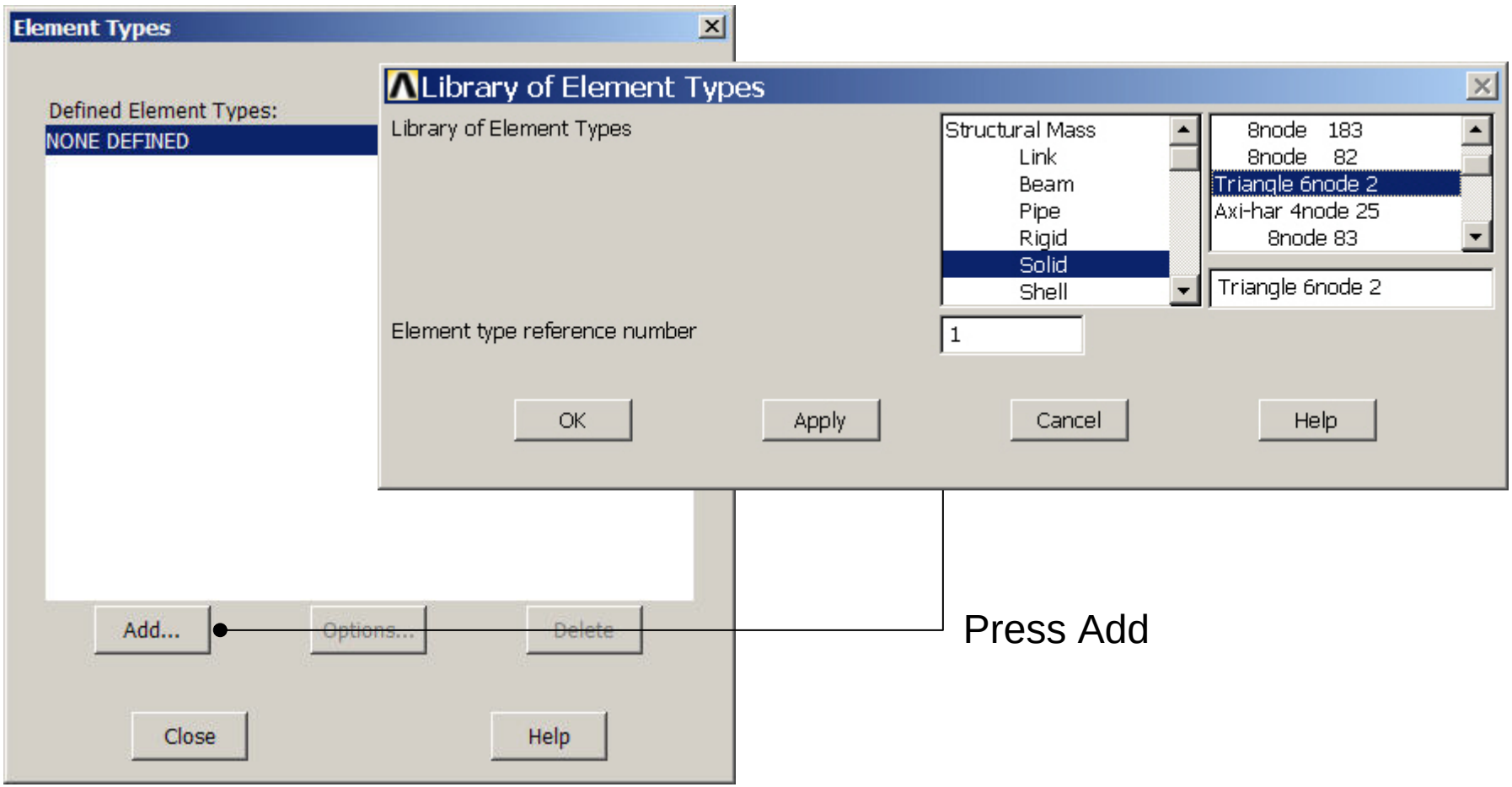
Press OK

Example – Areas



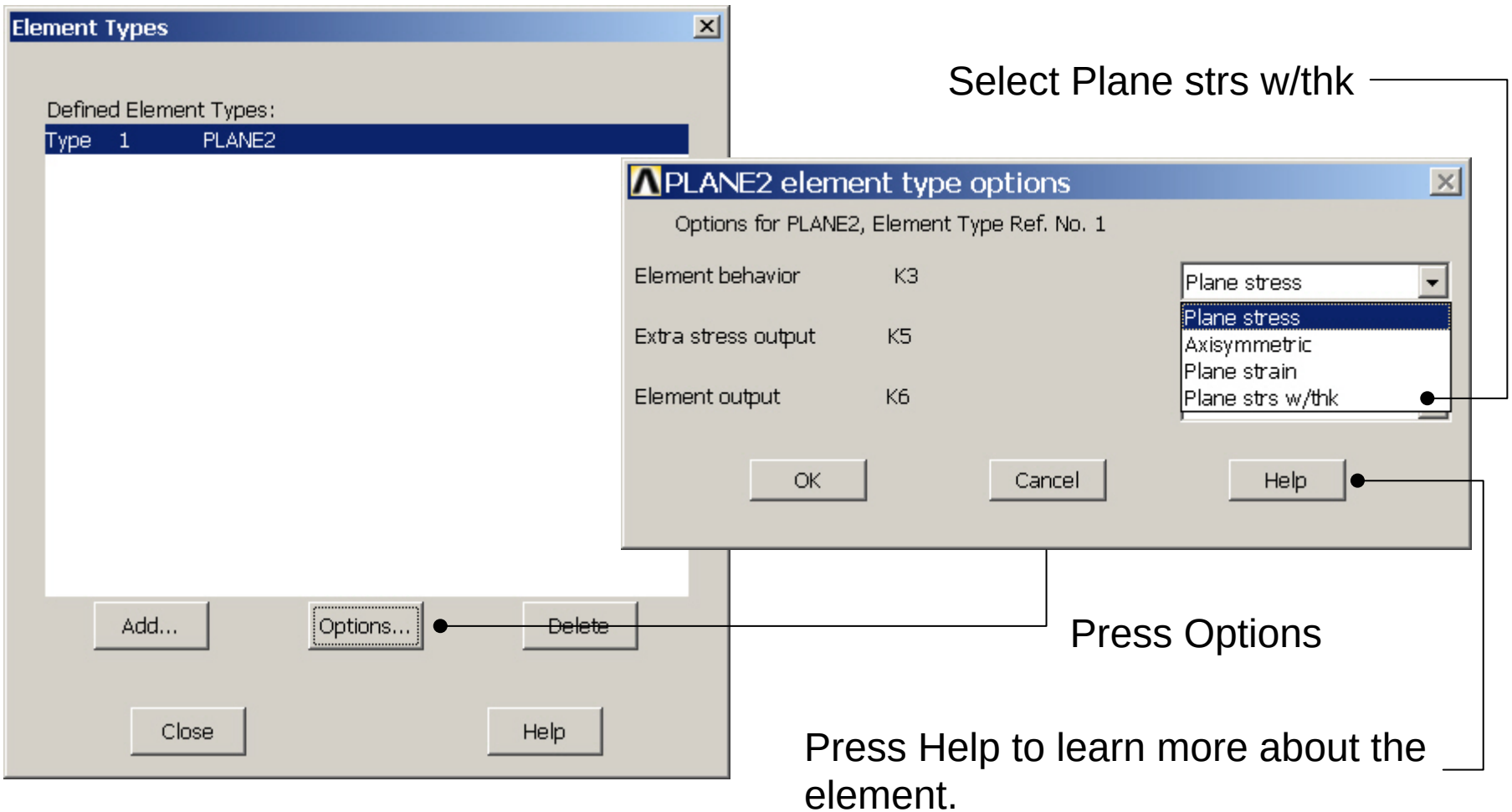
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



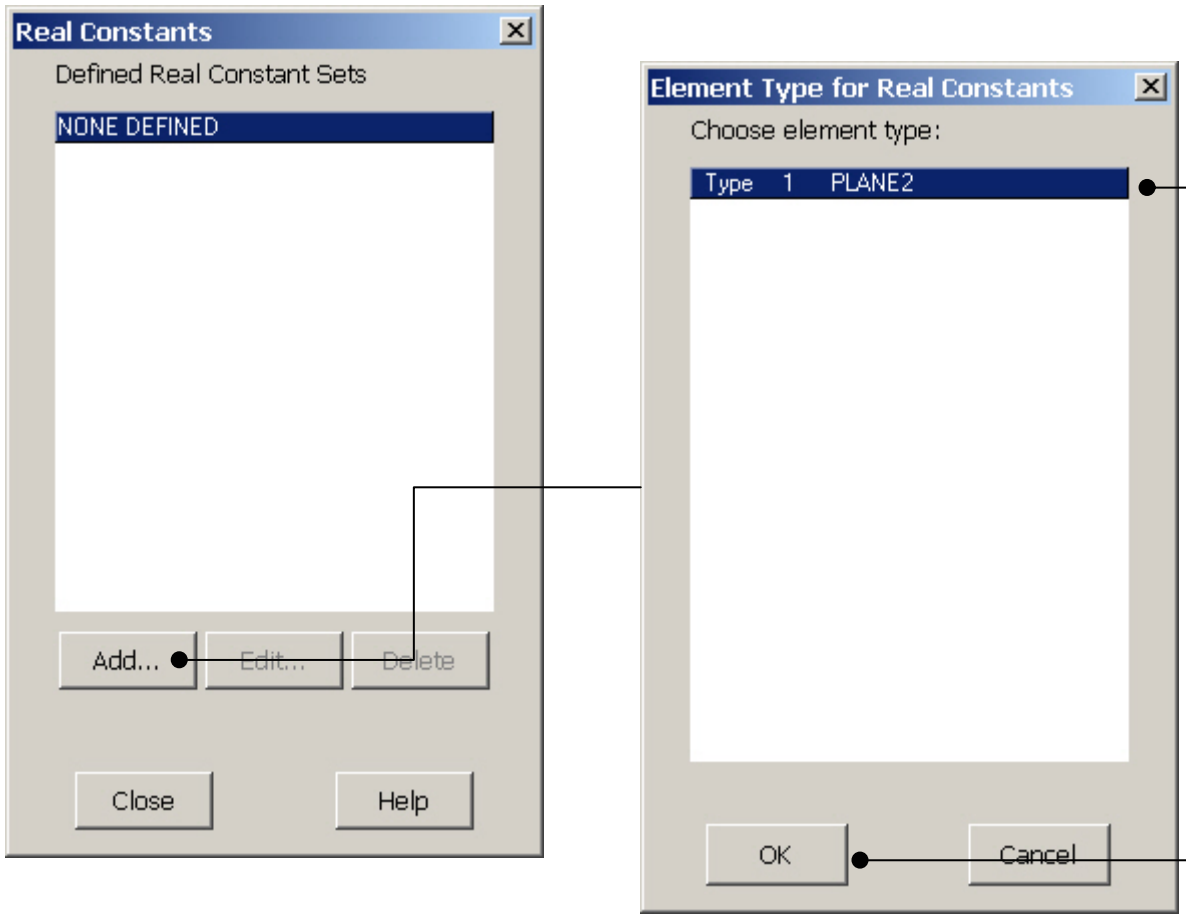
Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example – Real Constants

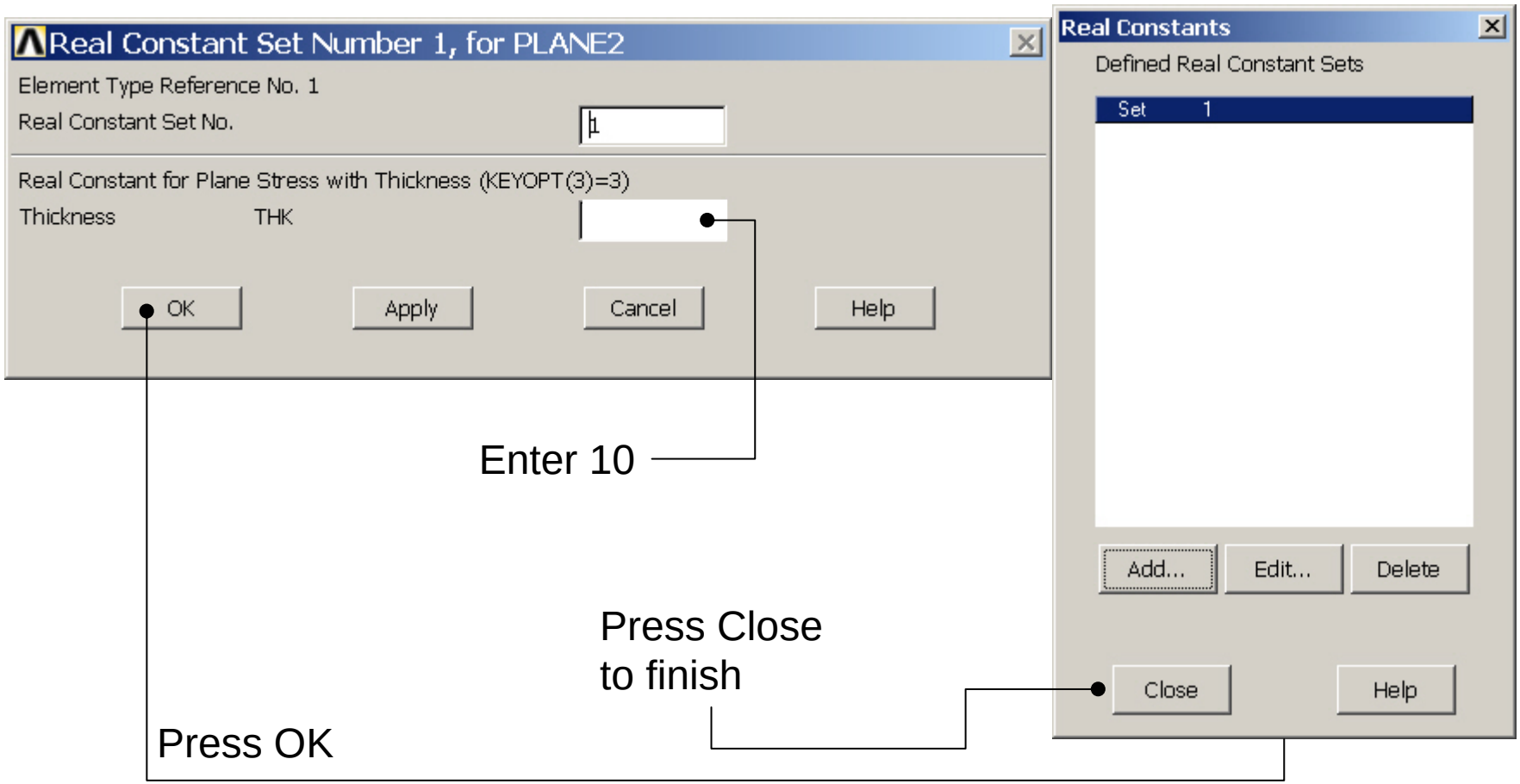
Preprocessor > Real Constants > Add



Place the cursor on the relevant element and press OK

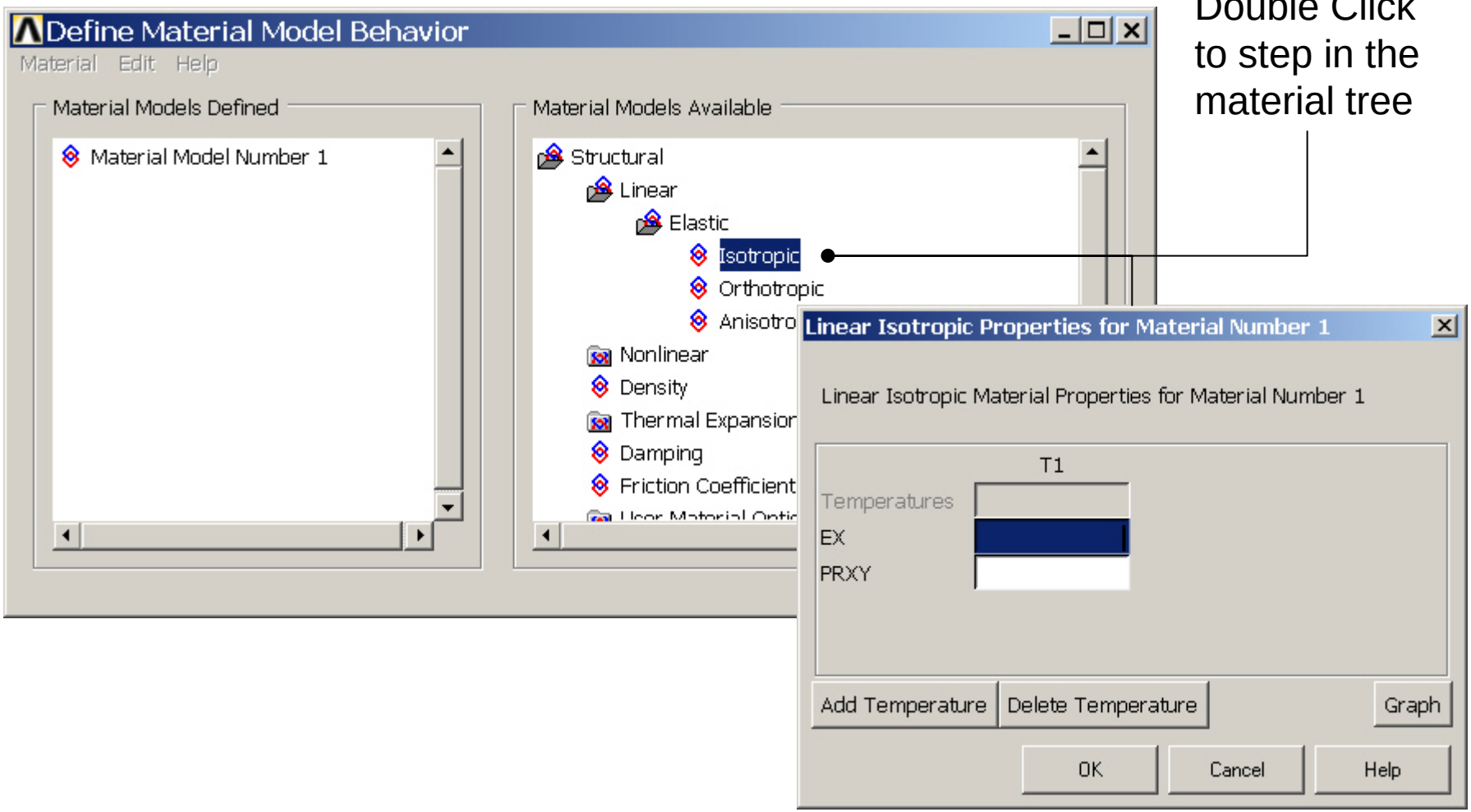
Example - Real Constants

Preprocessor > Real Constants > Add



Example - Material Properties

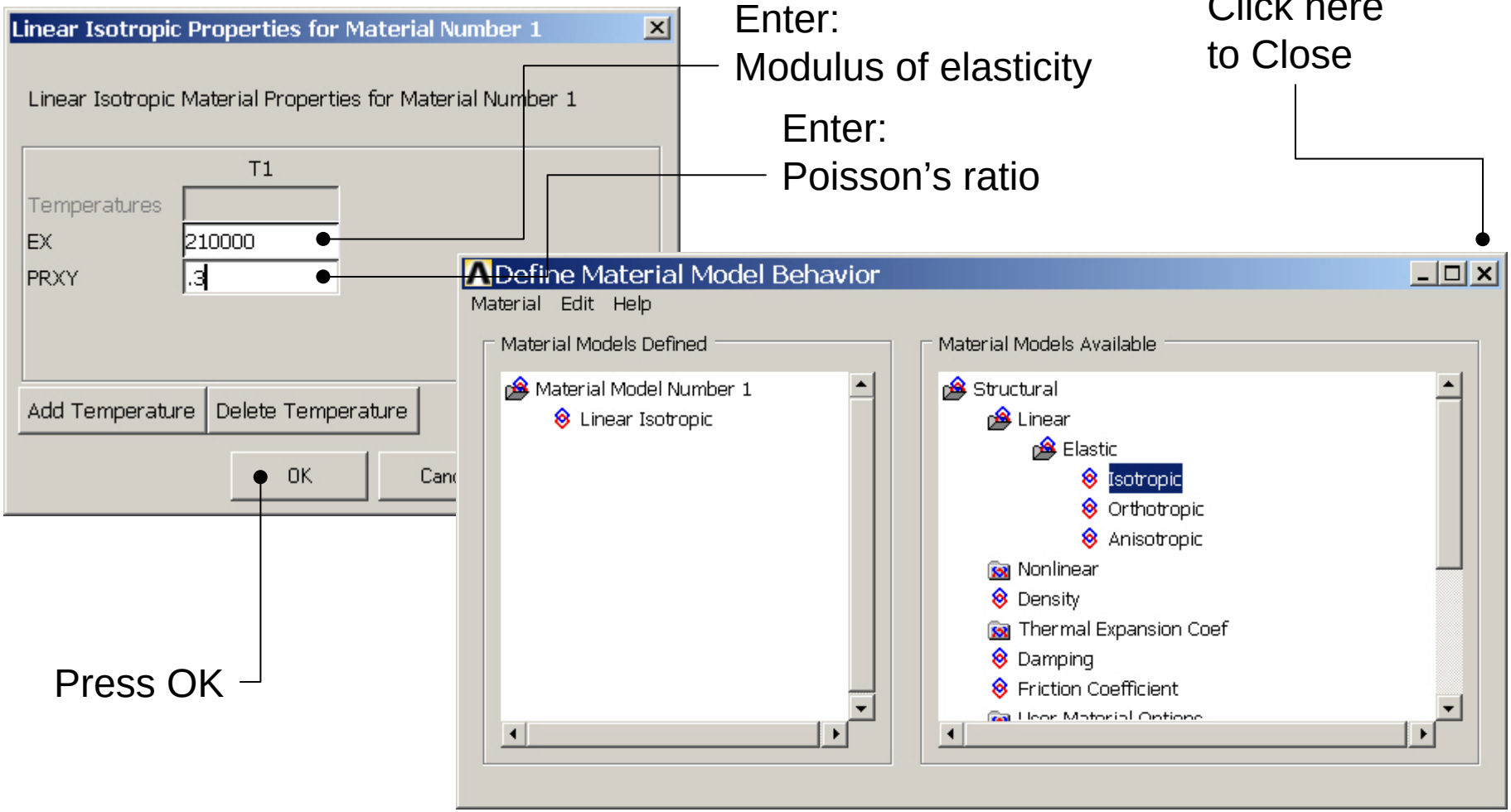
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

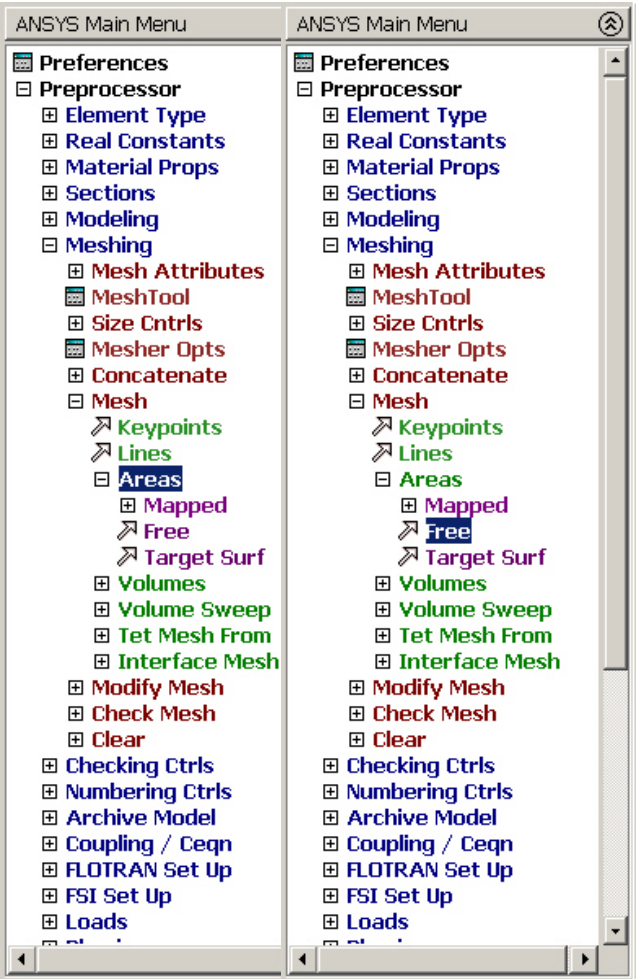
Example - Material Properties

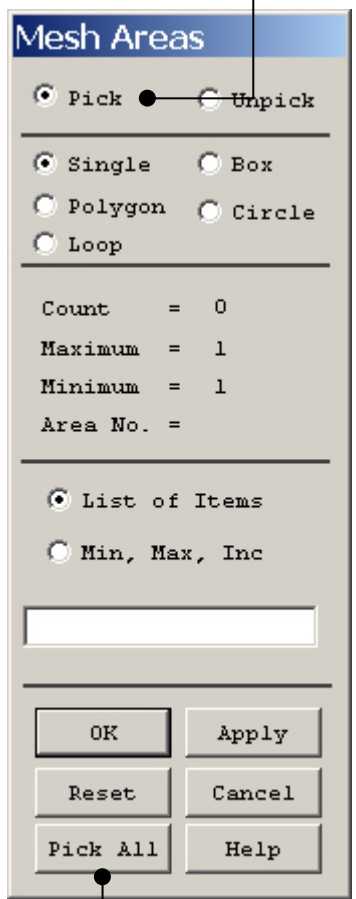
Preprocessor > Material Props > Material Models



Example – Free Meshing

Preprocessor > Meshing > Mesh > Areas > Free



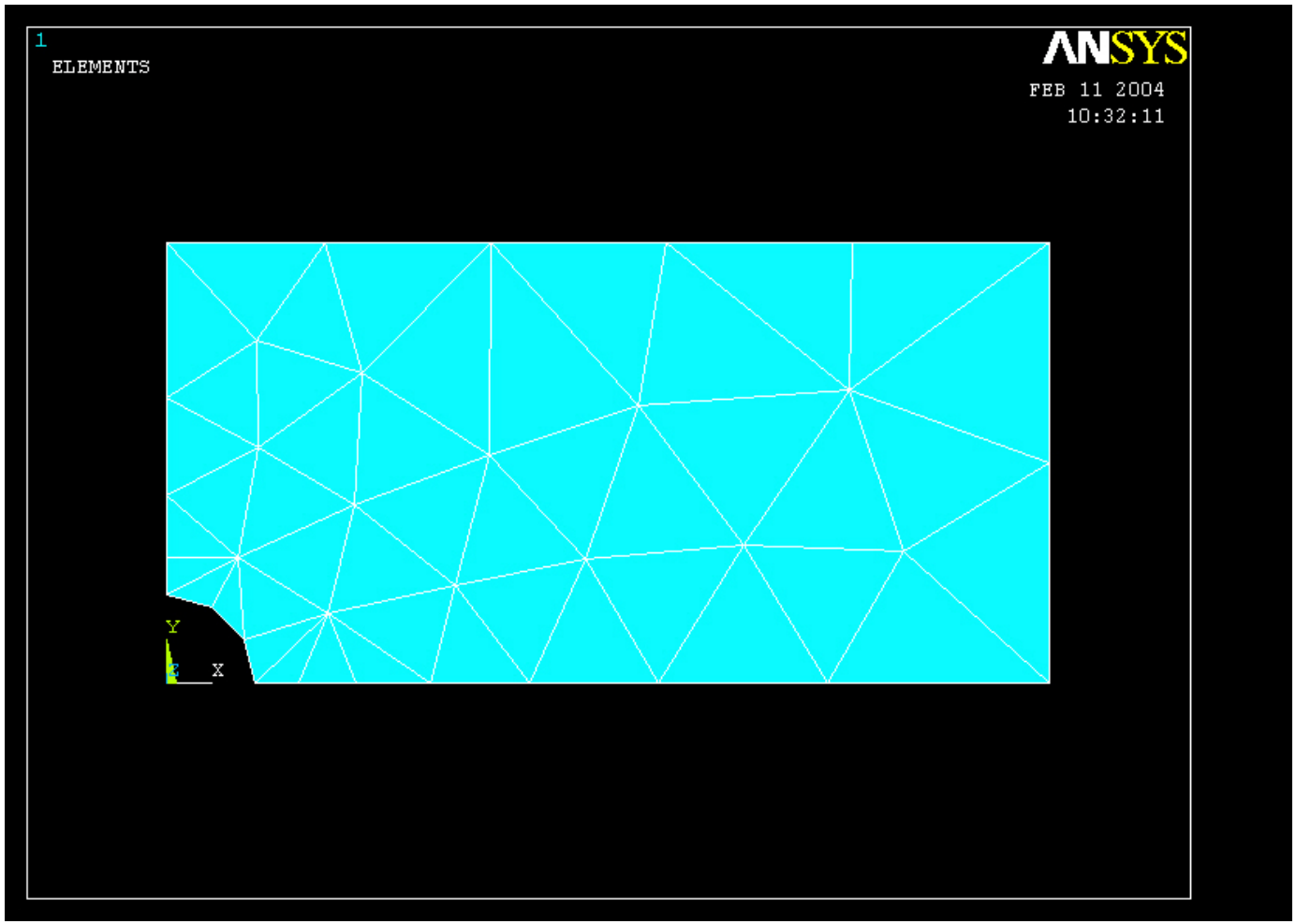


Select individual areas to be meshed

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

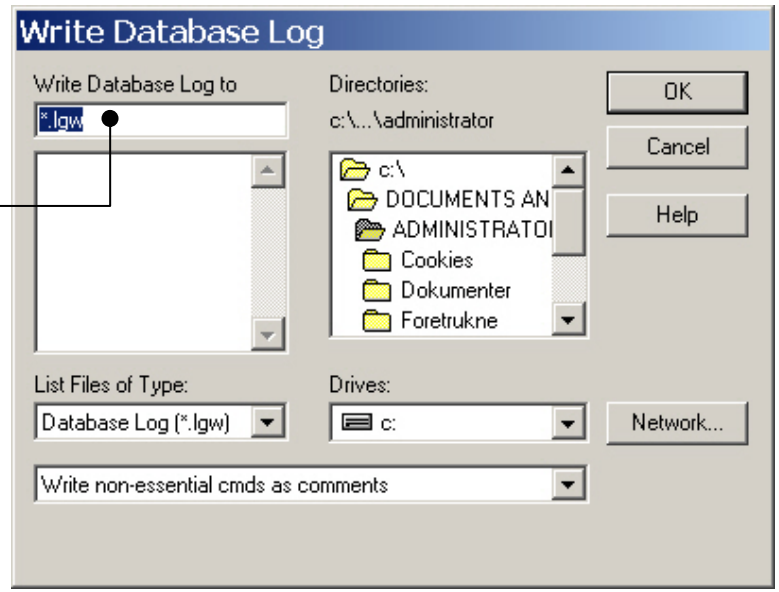
Example – Free Mesh



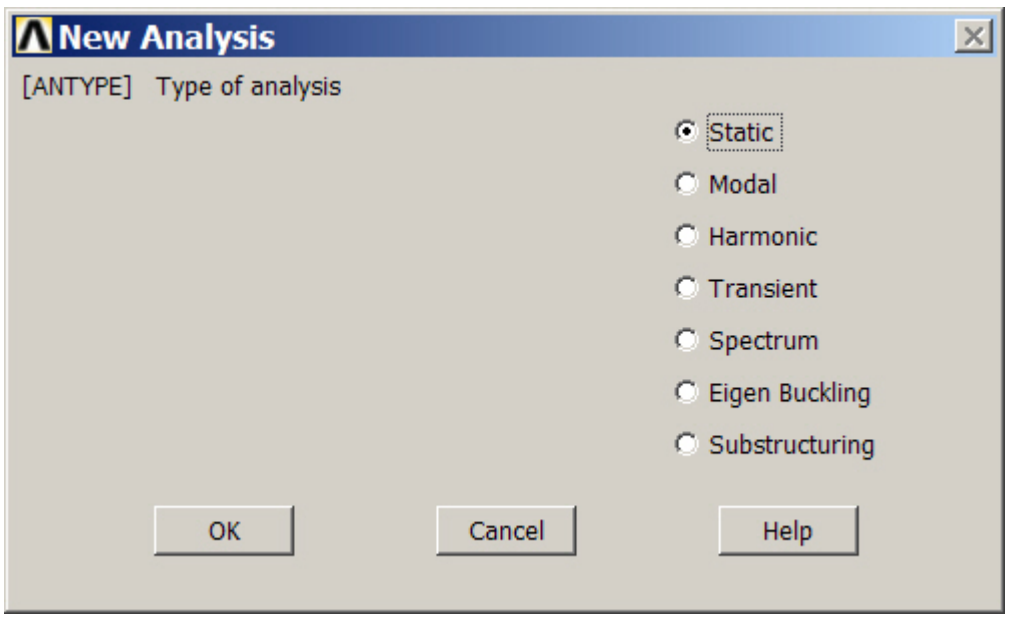
Example – Analysis Type

File > Write DB log file

Enter “example0240.lgw”

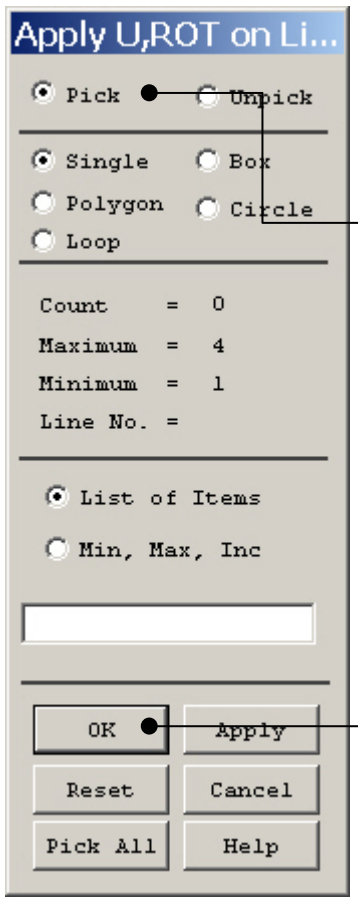


Solution > Analysis Type > New Analysis



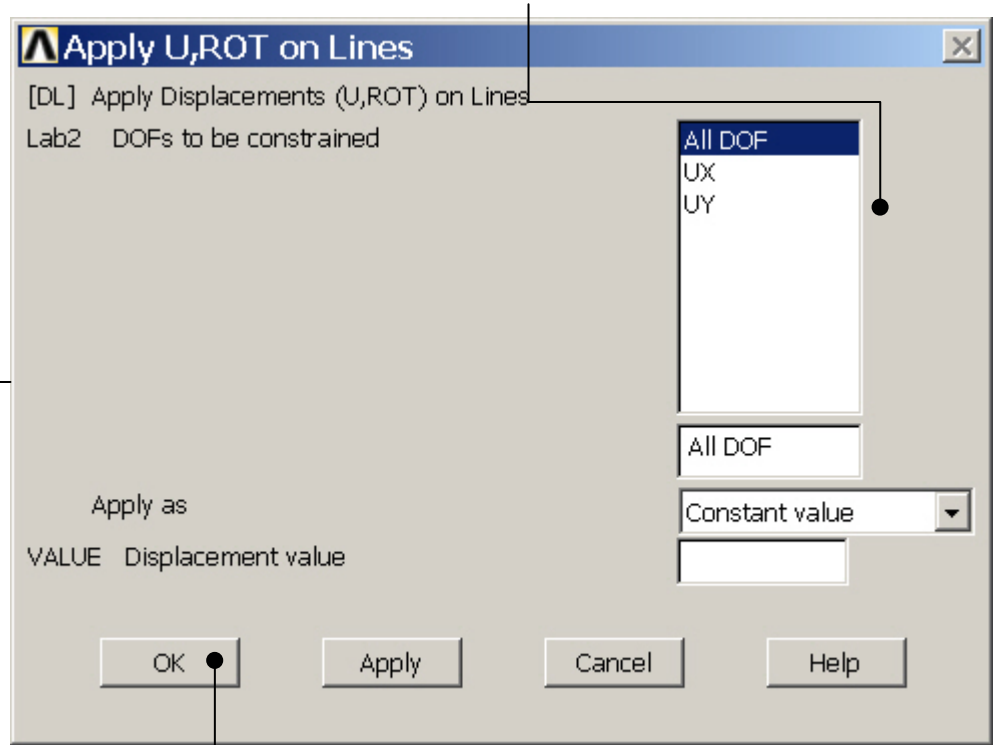
Example – Define Loads

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



Select the bottom straight line

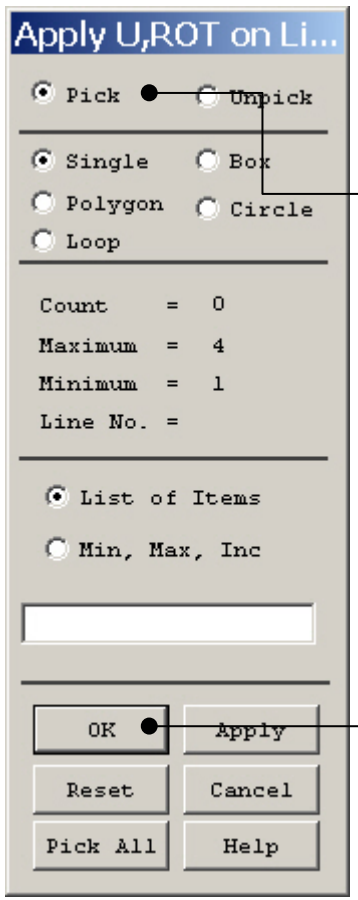
Select UY to fix the beam in the y-direction



Press OK

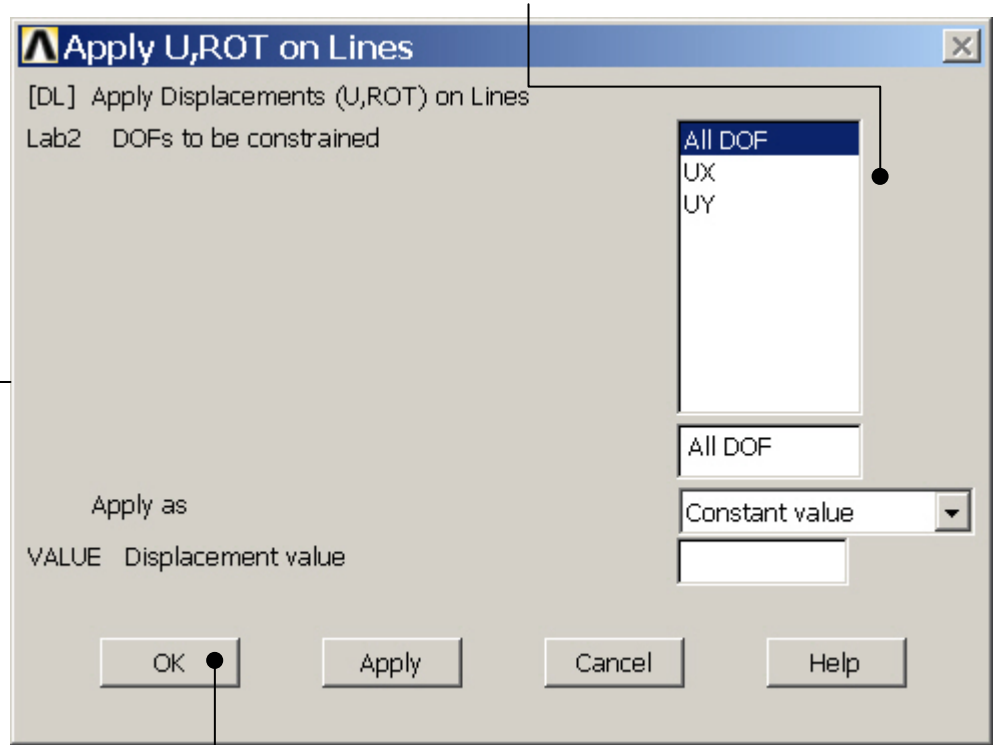
Example – Define Loads

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



Select the left straight line

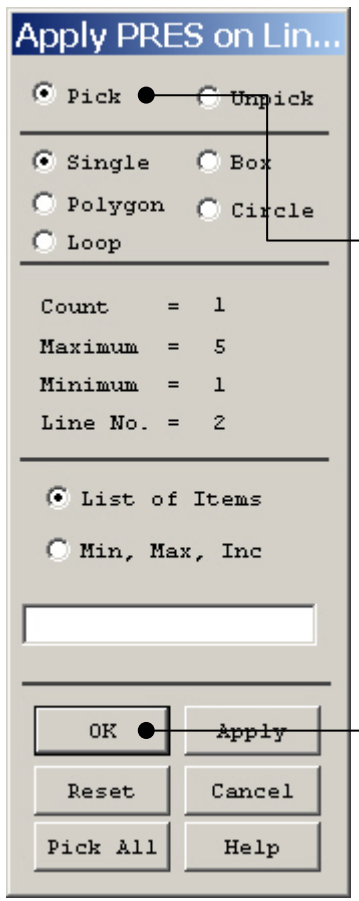
Select UX to fix the beam in the x-direction



Press OK

Example – Define Loads

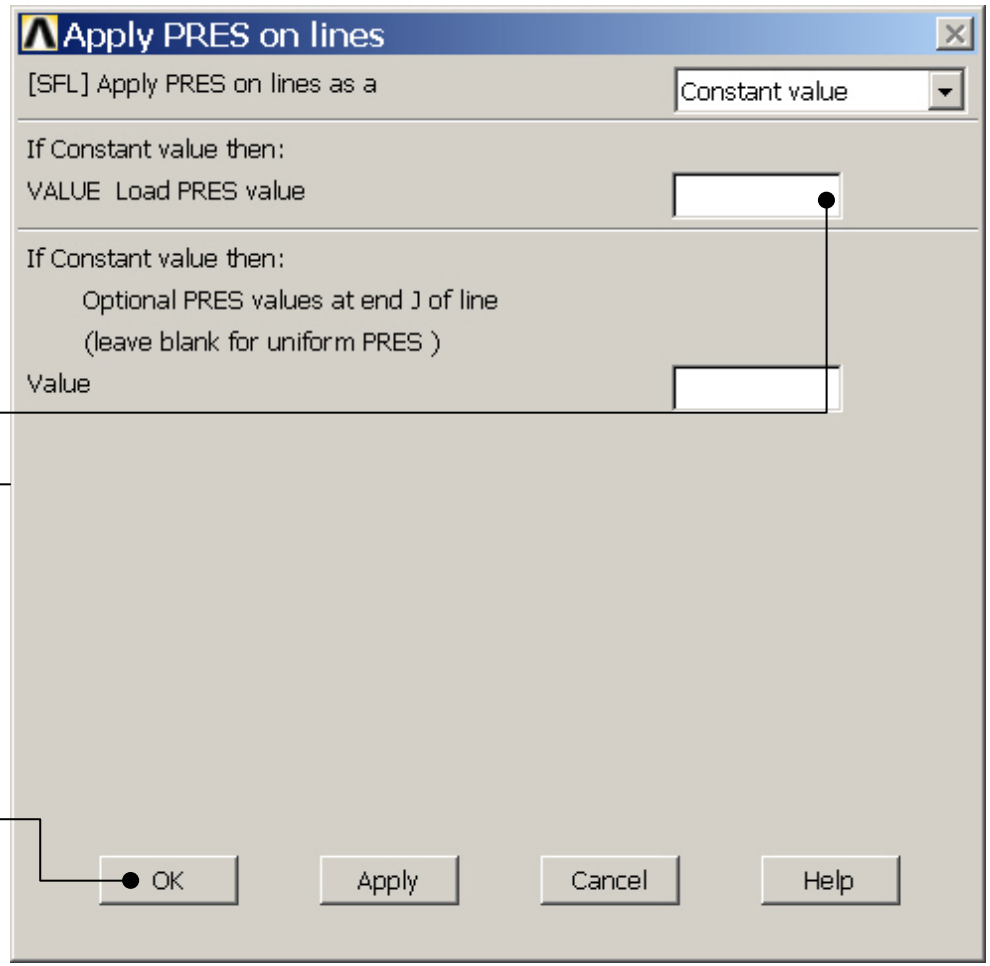
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the right straight line

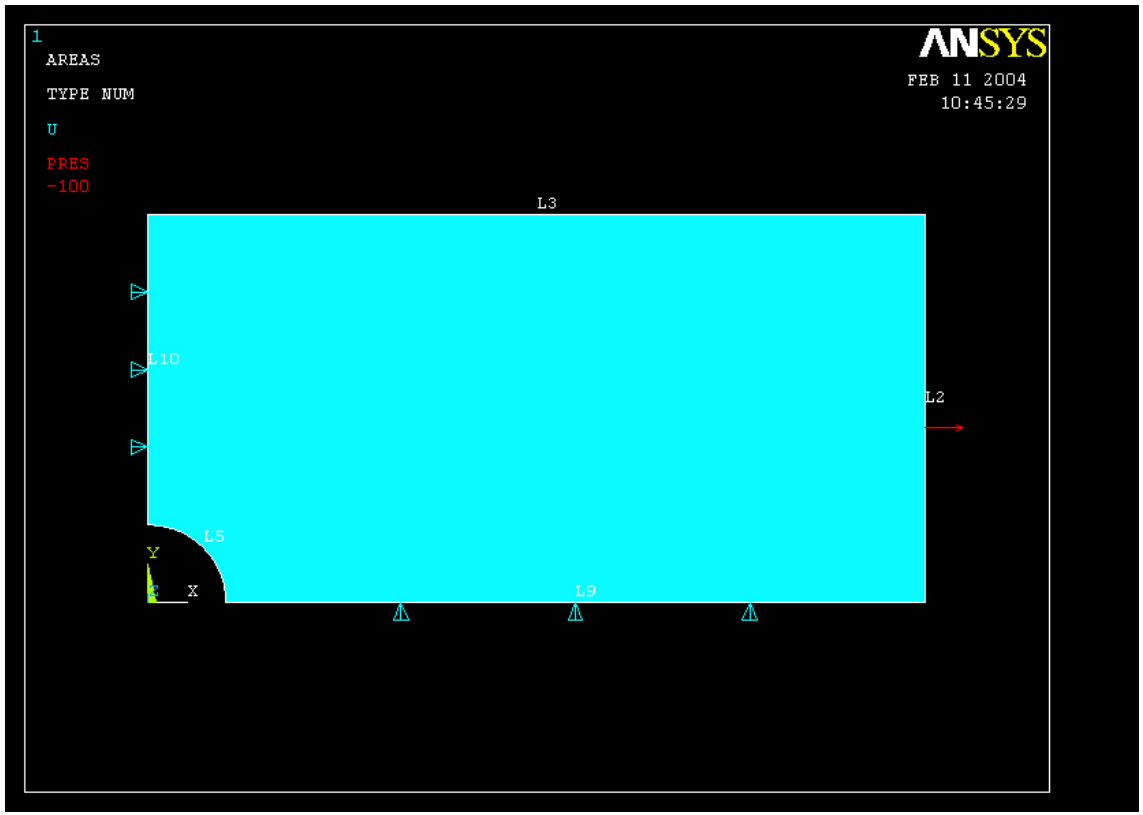
Enter -100

Press OK to finish

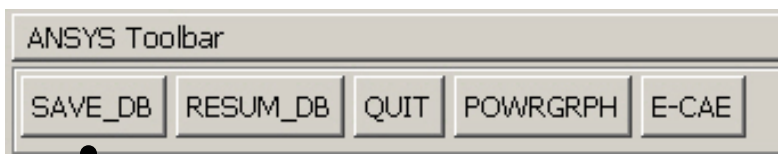


Note: Pressure acts normal and inward to a surface

Example - Save



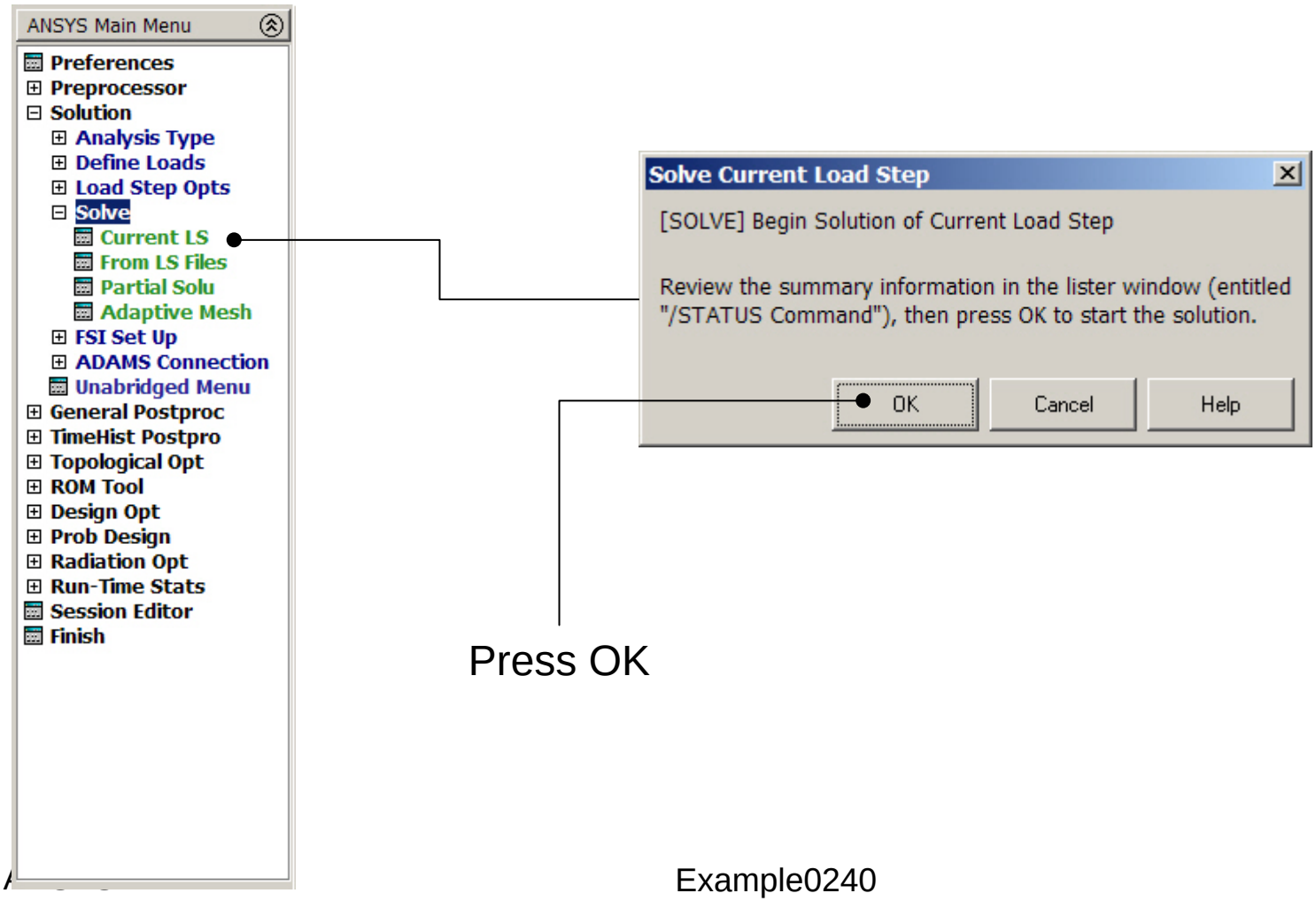
Display of Analysis model



Save the model

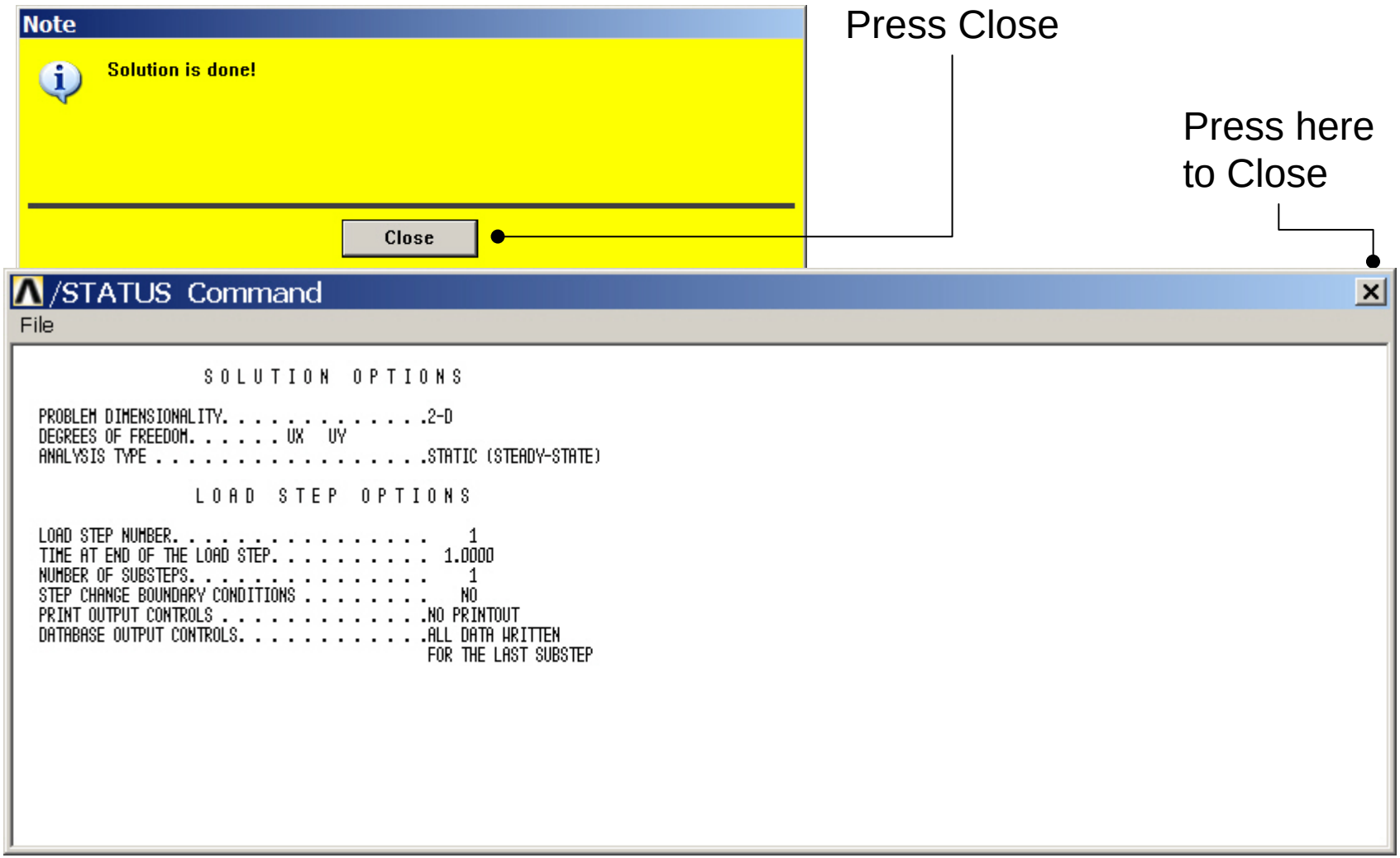
Example - Solve

Solution > Solve > Current LS



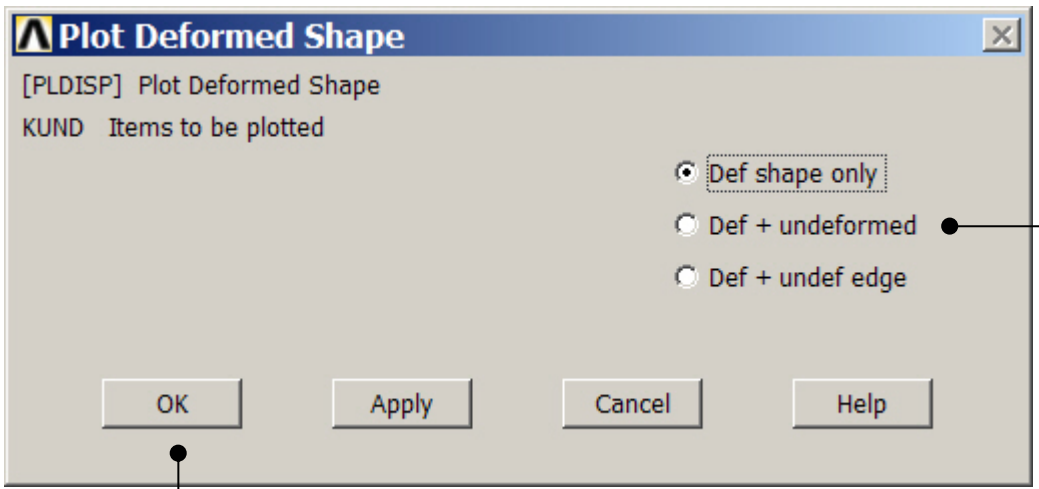
Press OK

Example - Solve



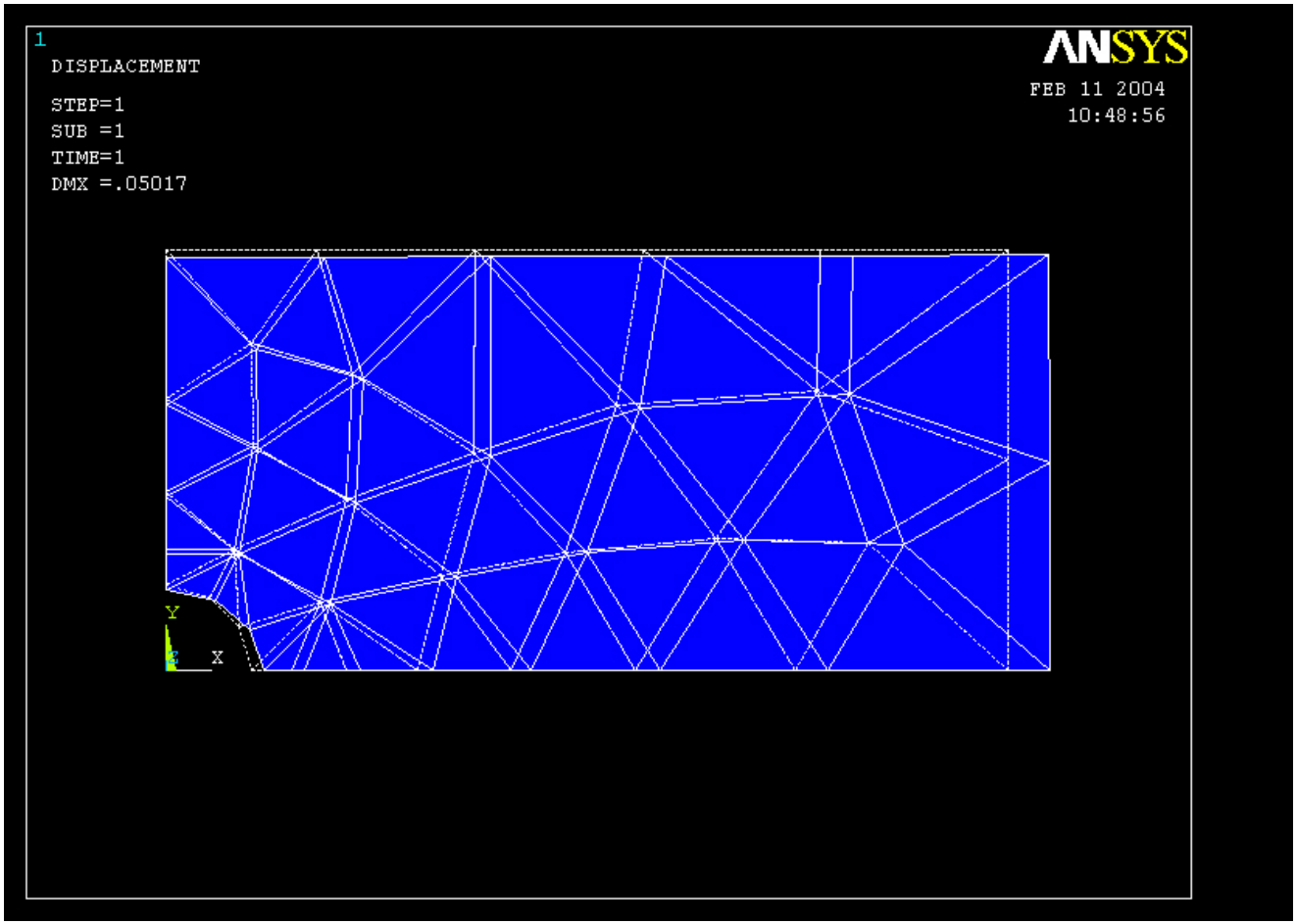
Example - PostProcessing

General Postproc > Plot Results > Deformed Shape



Select “Def+undeformed” and Press OK

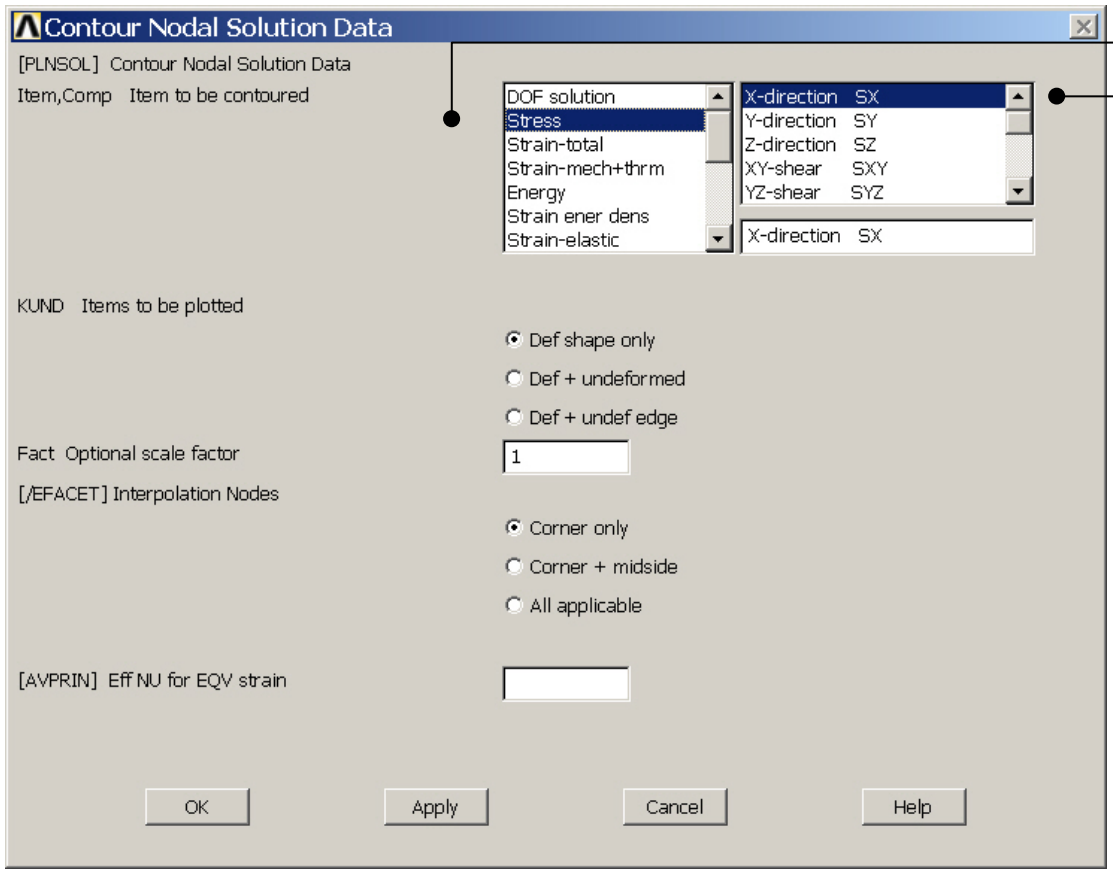
Example - PostProcessing



Read Maximum displacement: DMX

Example – Contour Plot

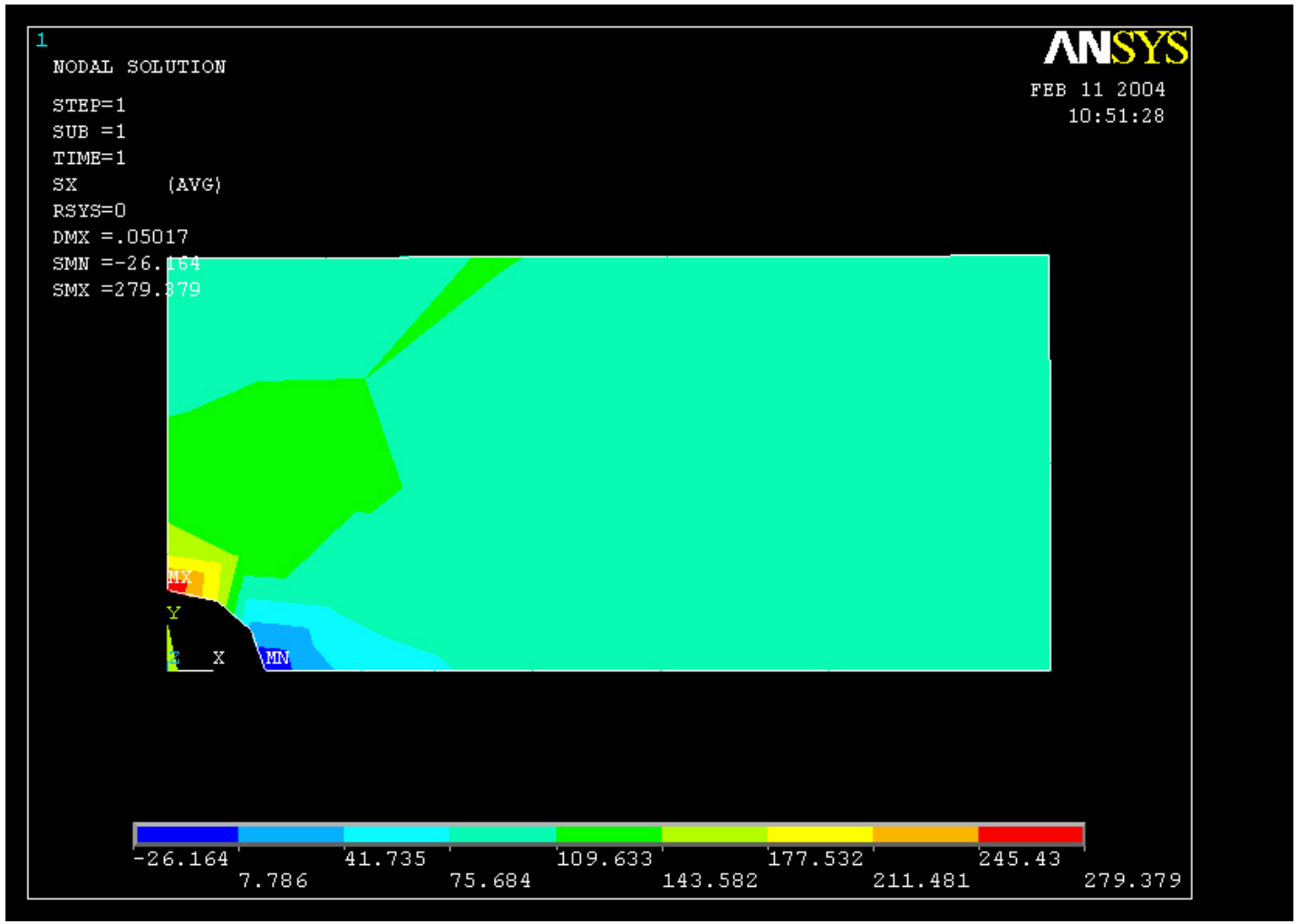
General Postproc > Plot Results > Contour Plot > Nodal Sol



Select Stress

Select SX for stresses in x-direction

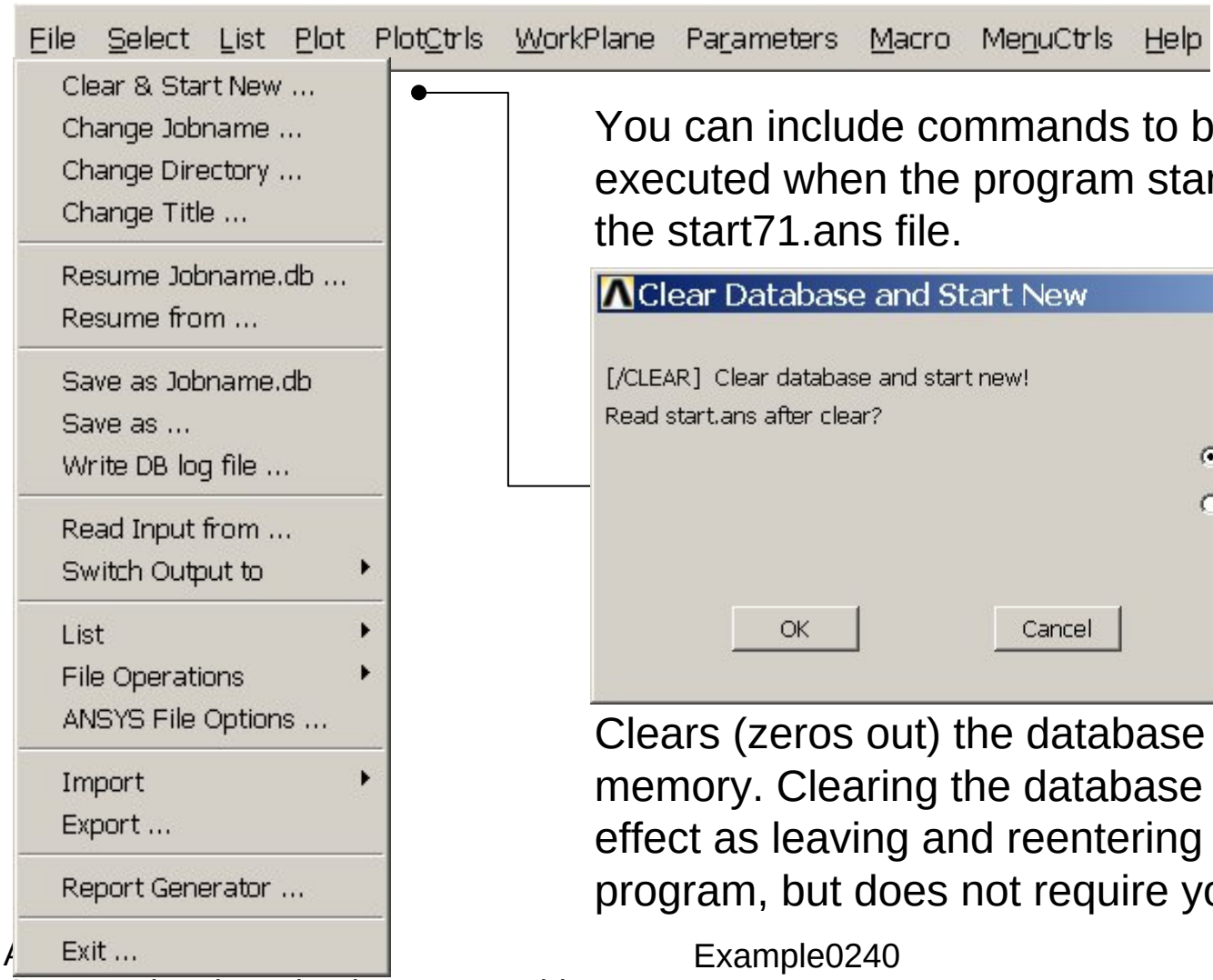
Example – Contour Plot



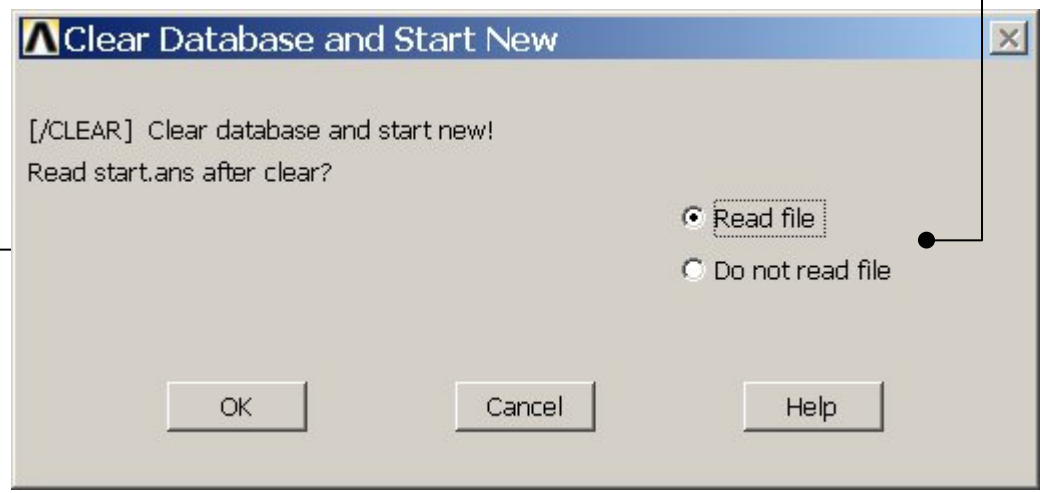
Example – Comments/Questions

- 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.