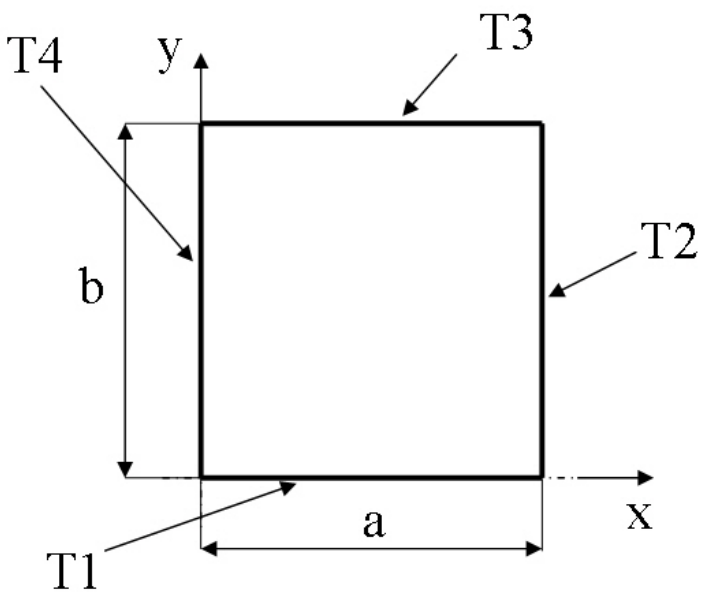
Course in ANSYS

Example0600

Example – Temperature in a plate



Objective:
Solve for the temperature distribution within the rectangular plate, based on the specified temperatures on the plate edges, and the plate dimensions.

Tasks:

How should this be modelled?

Topics:

Element type, thermal analysis, modeling, plot results, output graphics

E = 210000N/mm^2

ν = 0.3

Thermal conductivity = 401W/(m-K)

a = 10m

b = 20m

T1 = 100C

T2 = 100C

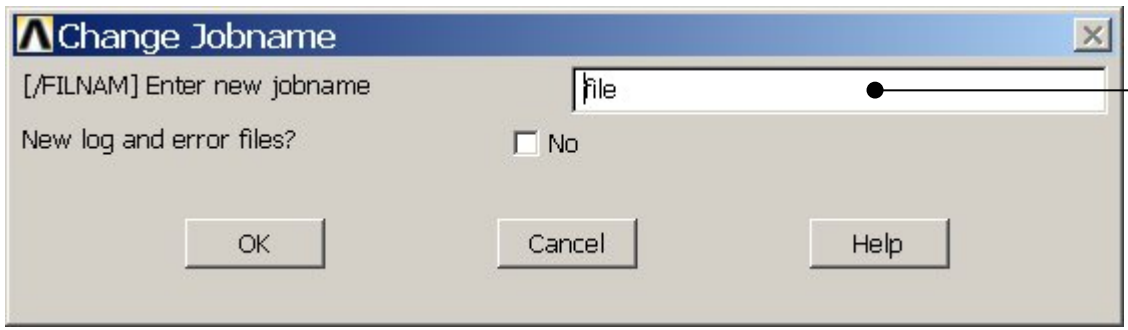
T3 = 200C

T4 = 100C

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0600 — Command line entry

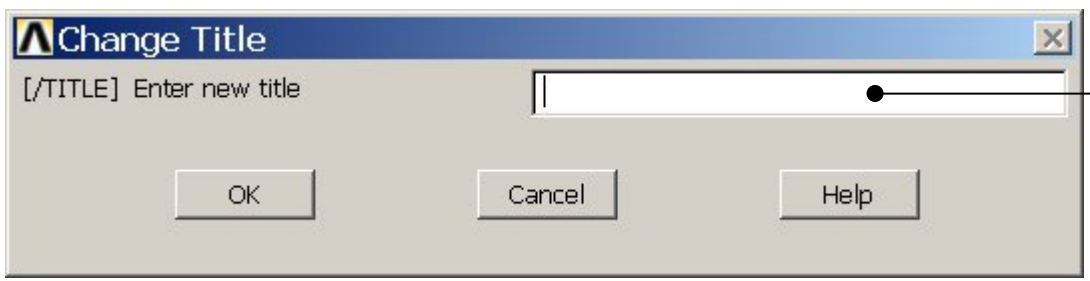


Enter: Example0600

Utility Menu > File > Change Title

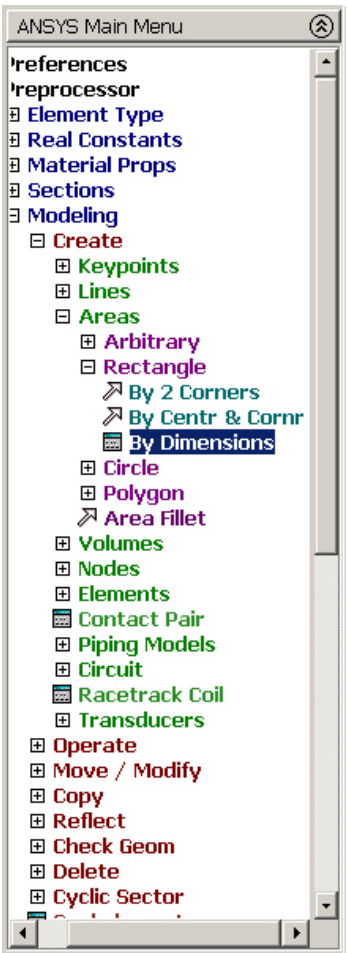
/title, Temperature in a plate

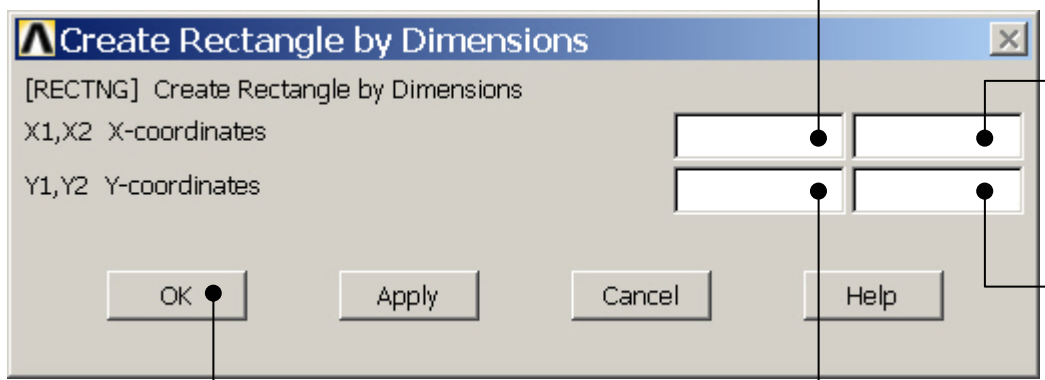
Enter: Temperature in a plate



Example – Areas Rectangle

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





Enter 0 or leave empty

Enter 10

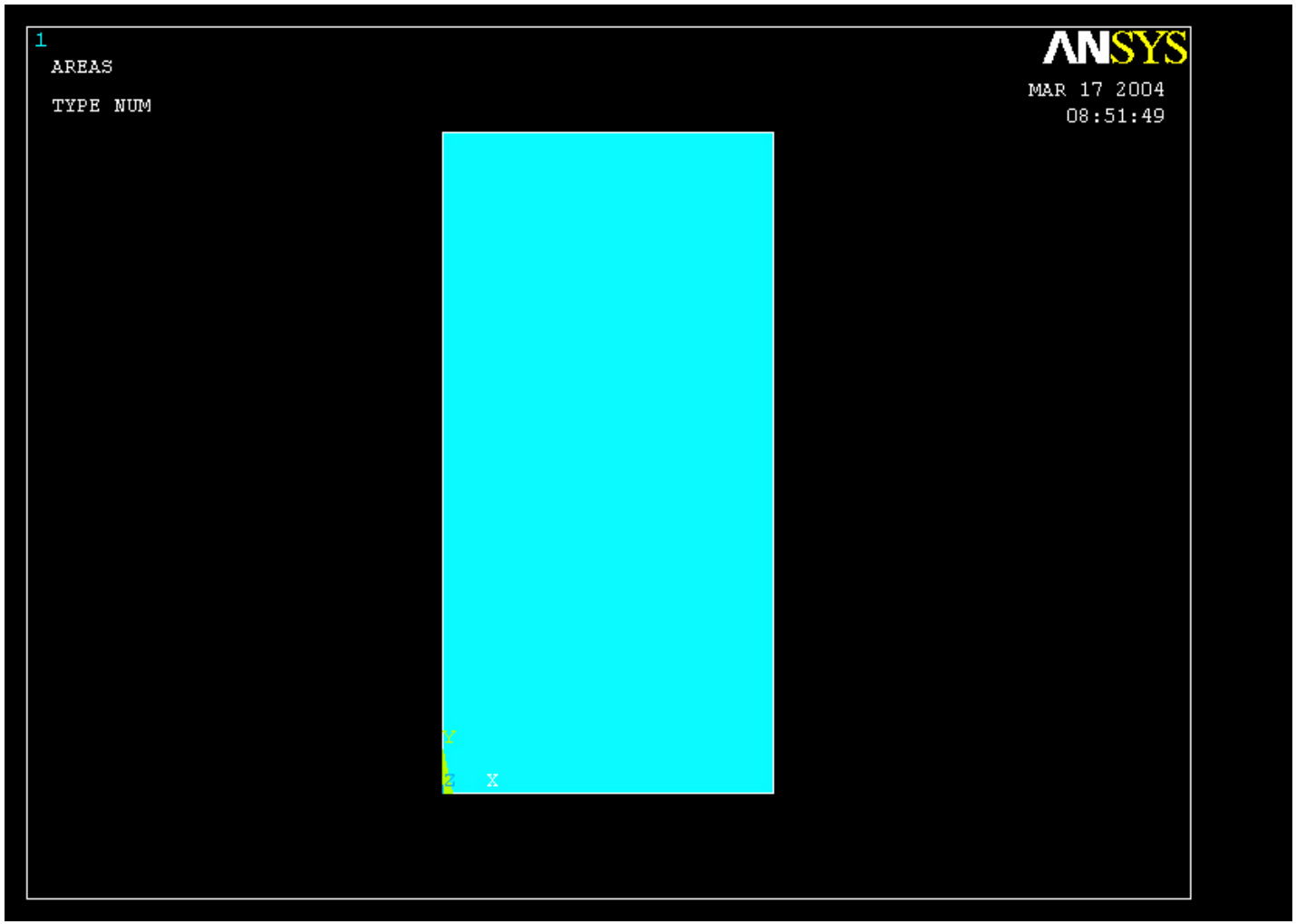
Enter 0 or leave empty

Enter 20

Press OK

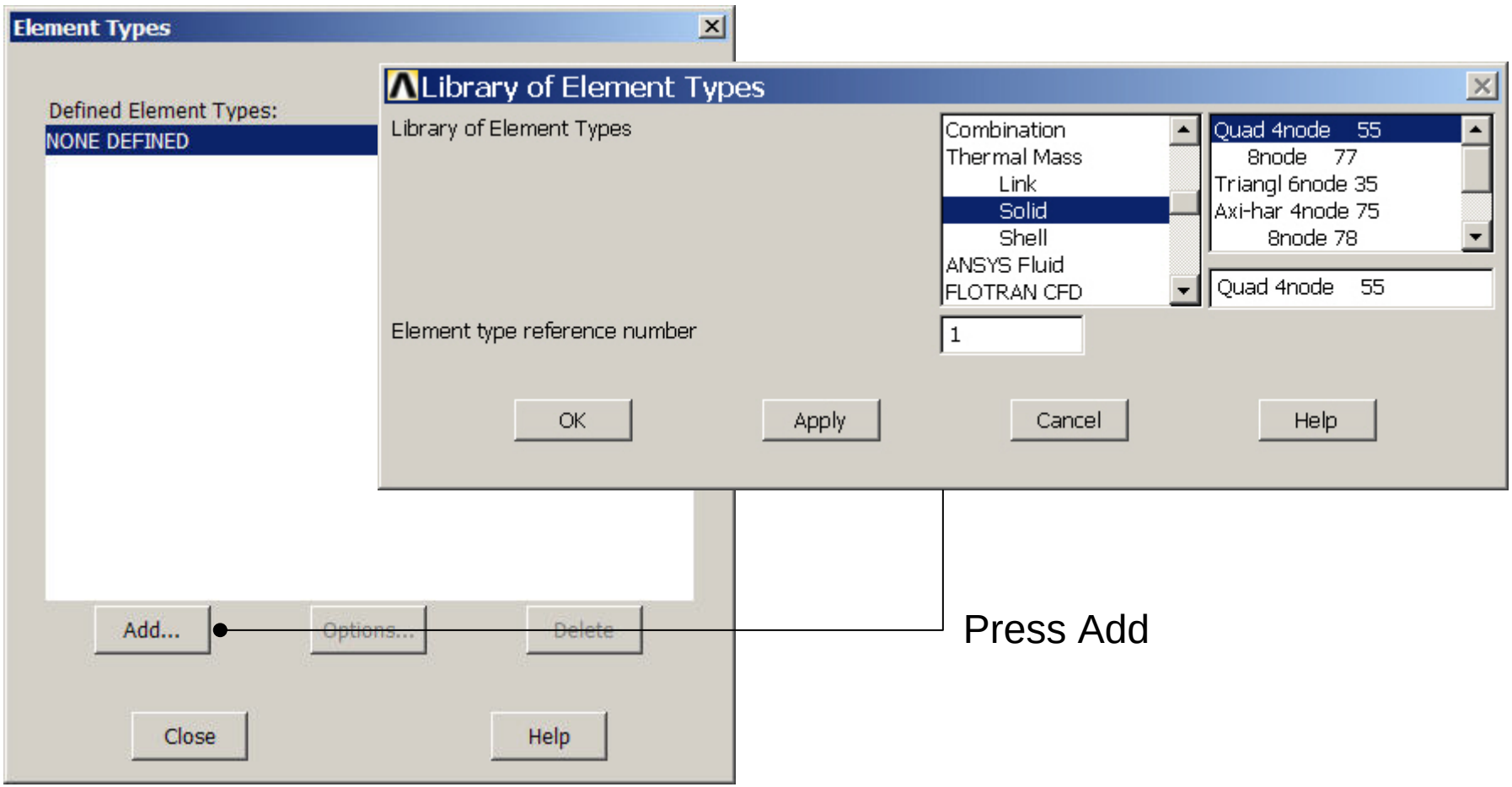
Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

Example – Areas Rectangle



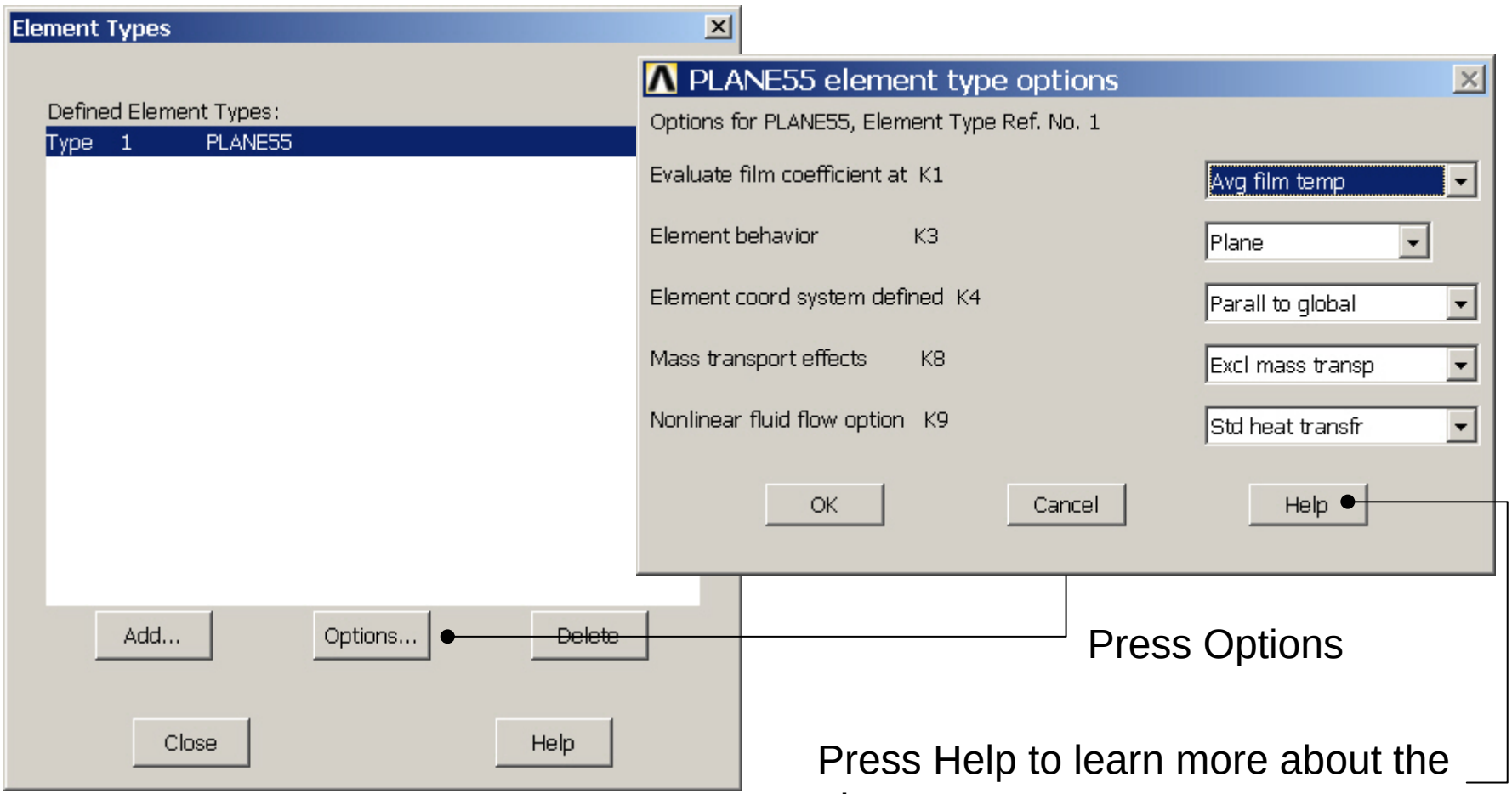
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete

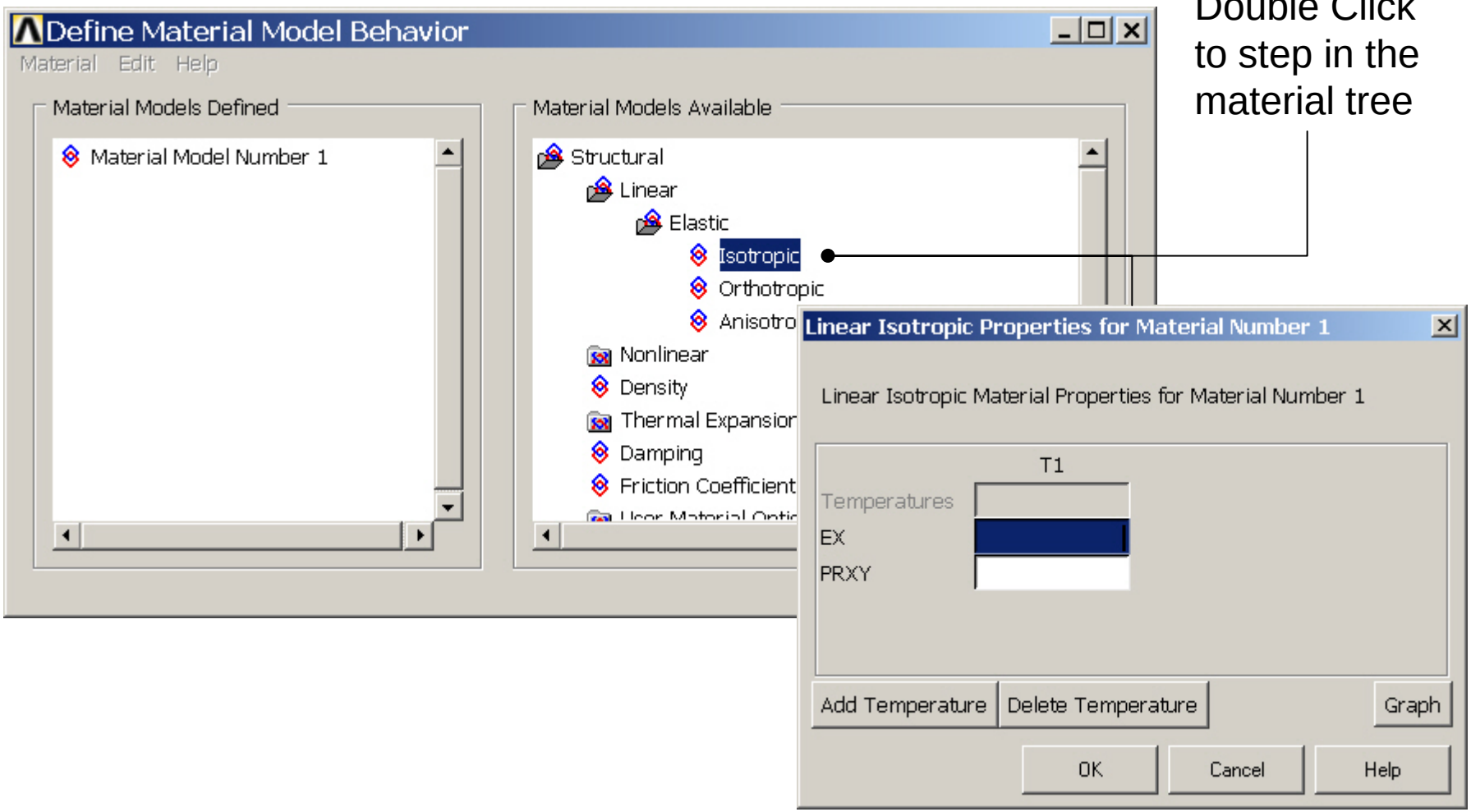


Press Options

Press Help to learn more about the element.

Example - Material Properties

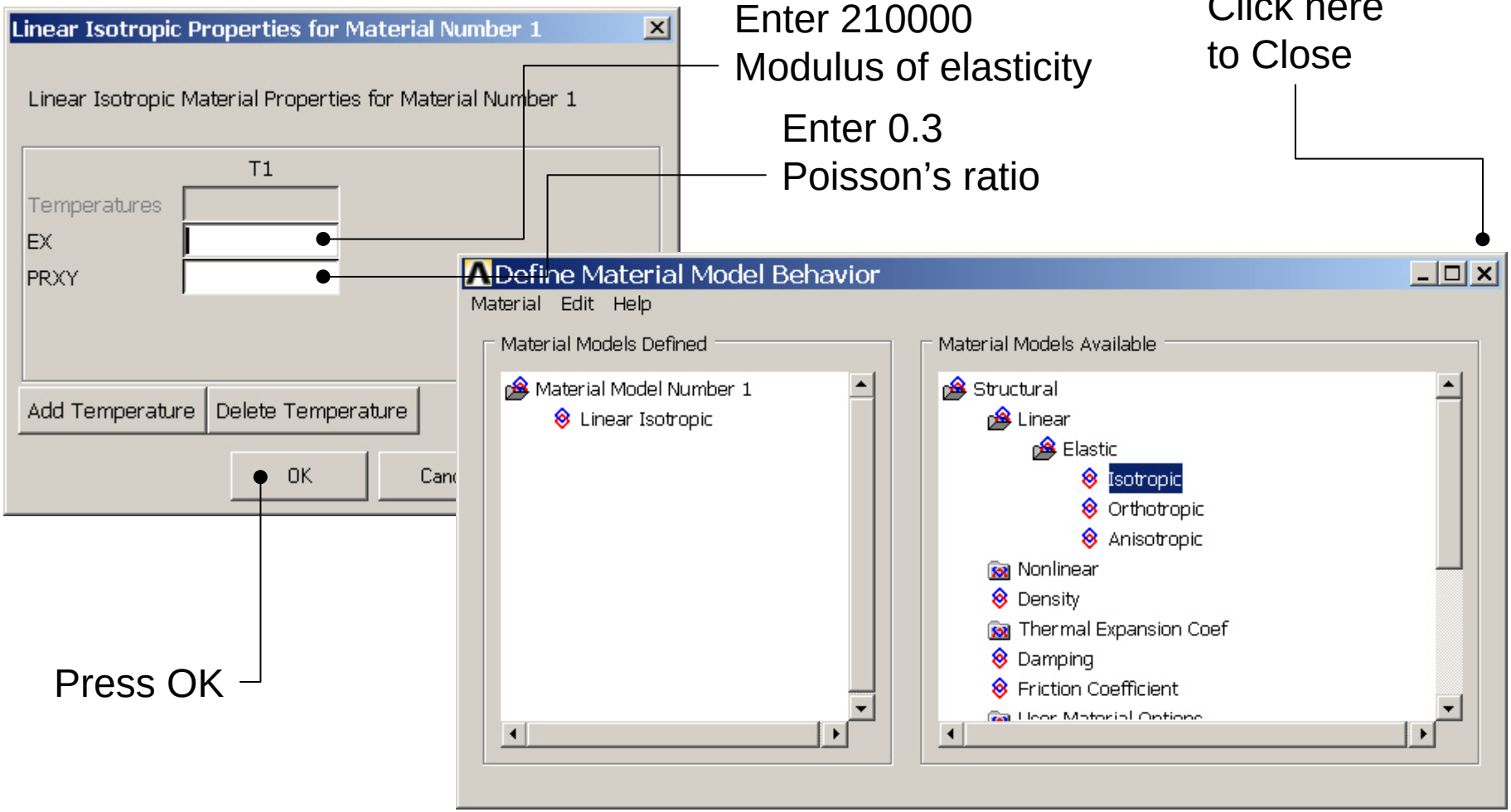
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

Example - Material Properties

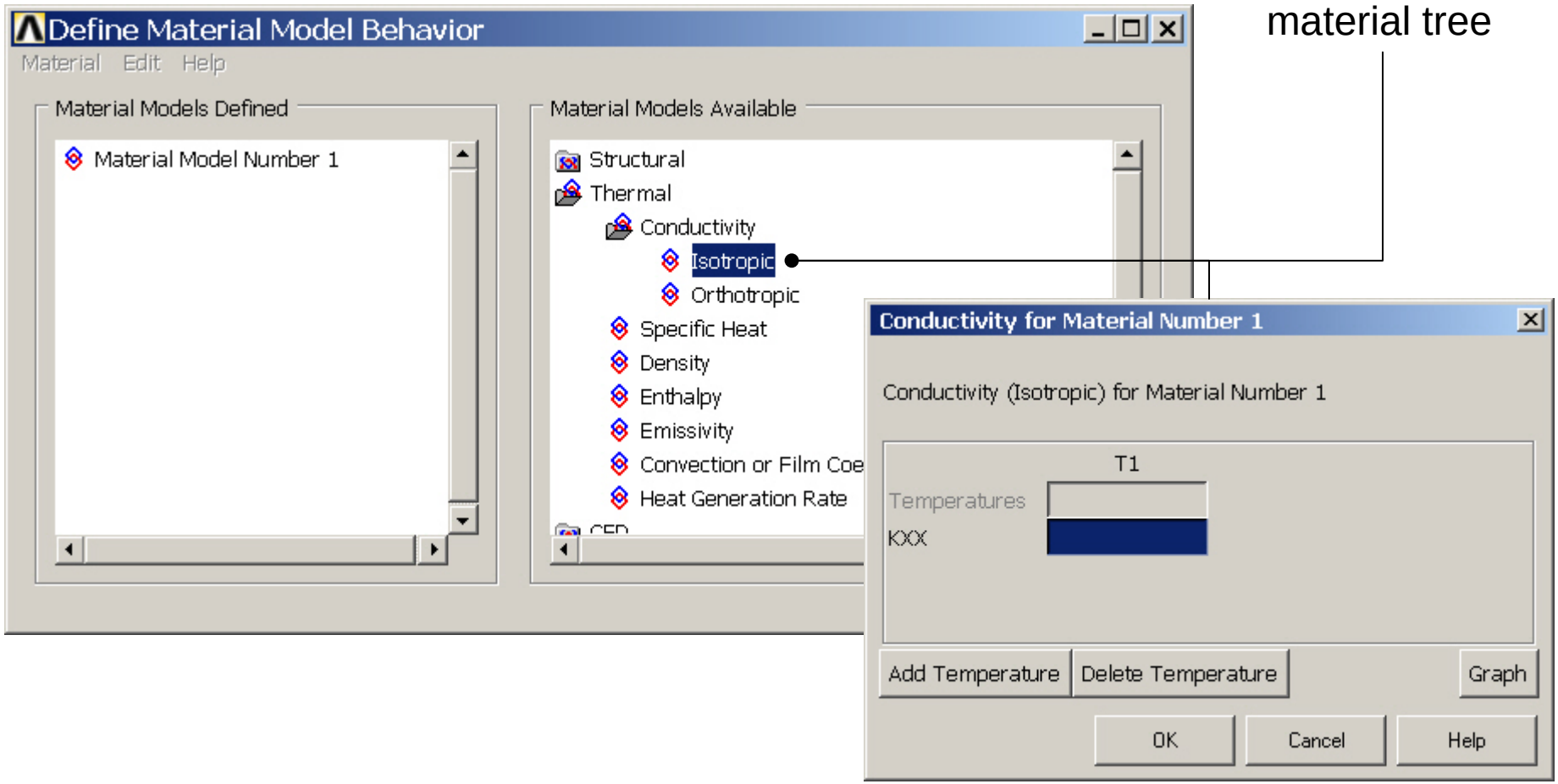
Preprocessor > Material Props > Material Models



Example - Material Properties

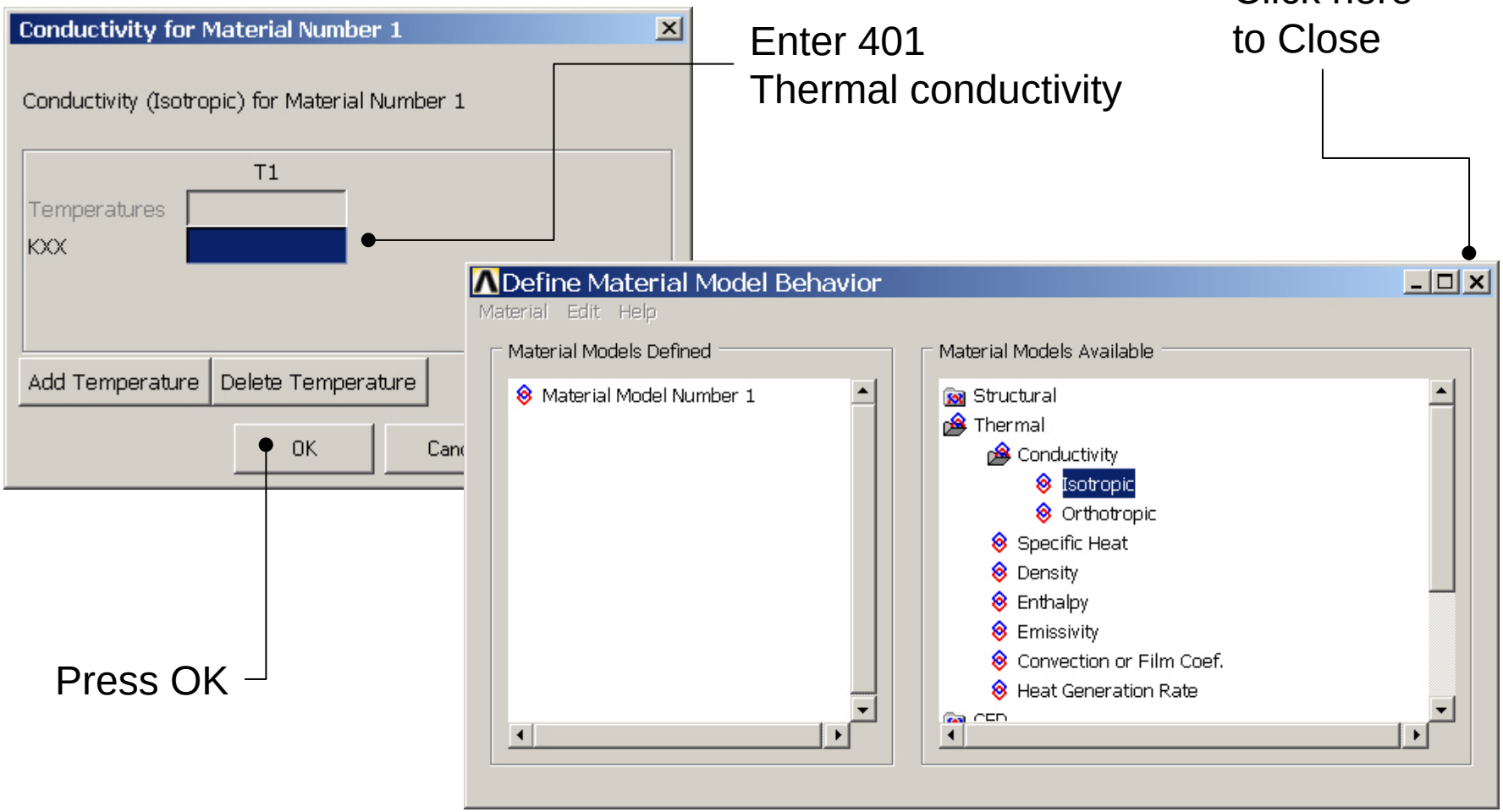
Preprocessor > Material Props > Material Models

Double Click to step in the material tree



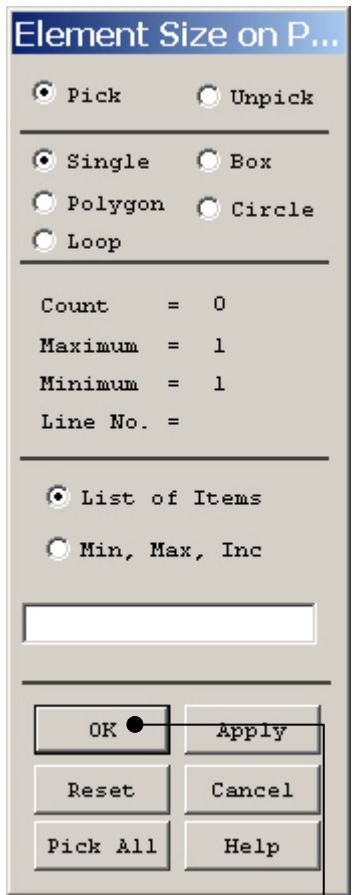
Example - Material Properties

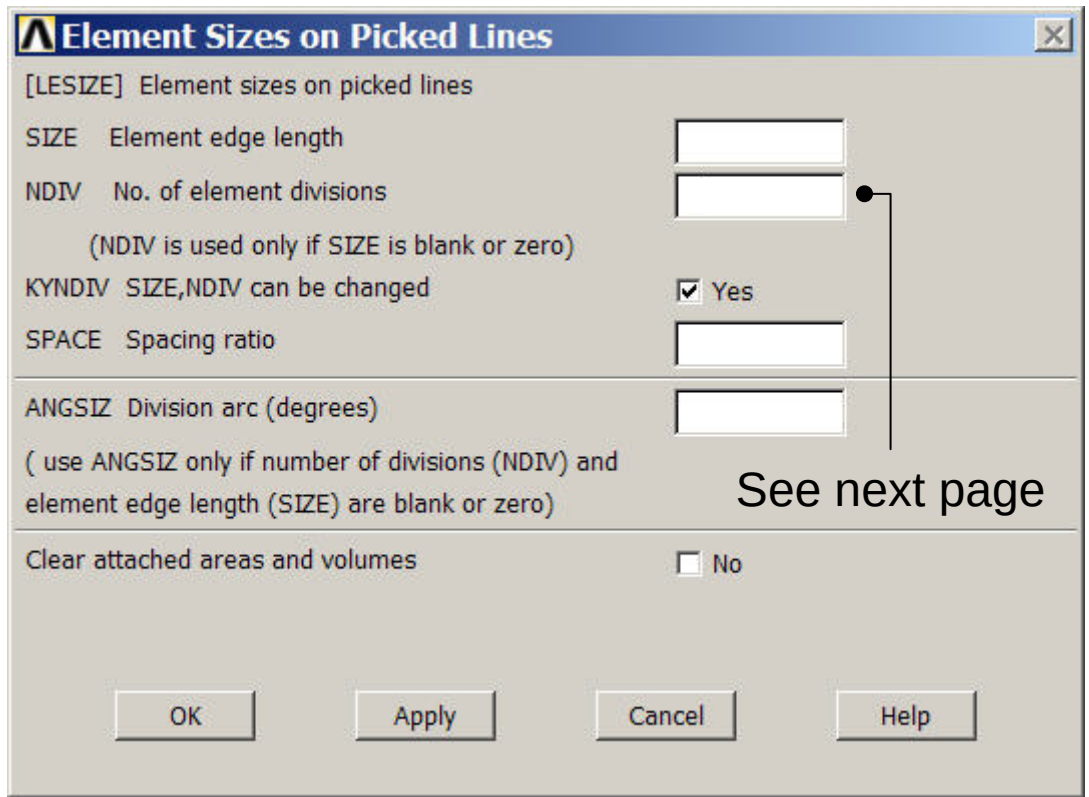
Preprocessor > Material Props > Material Models



Example - Meshing

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





Select/Pick Lines to specify mesh size for

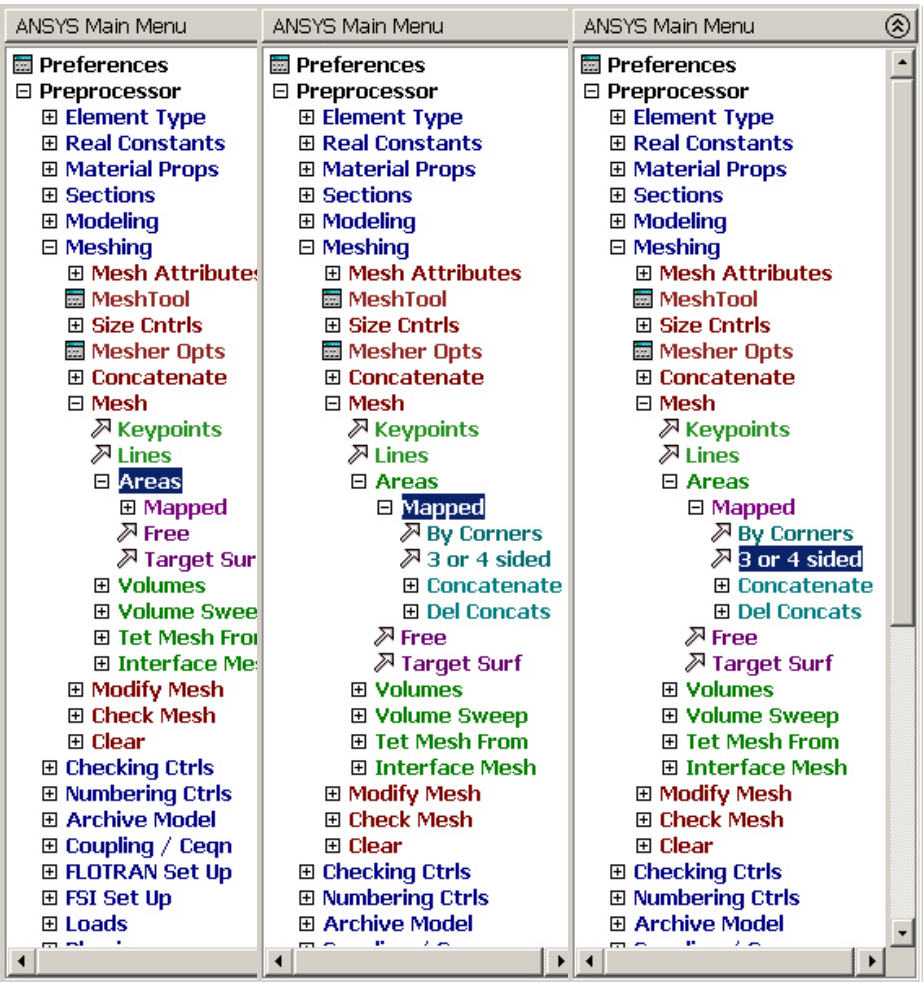
Pick the two longest lines

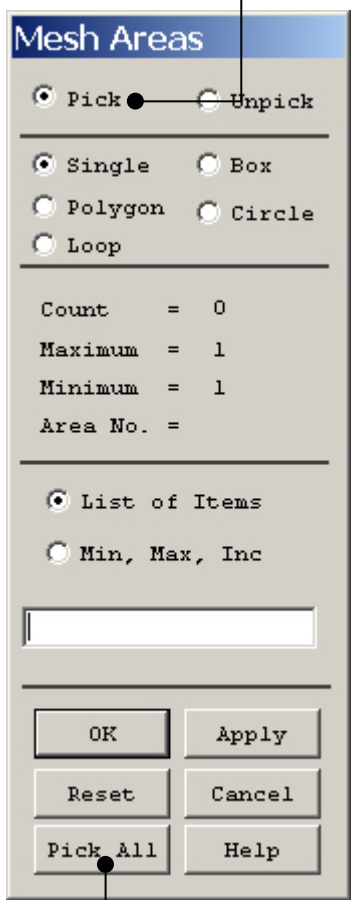
See next page

Press OK when finish with selection

Example - Meshing

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





Select individual areas to be meshed

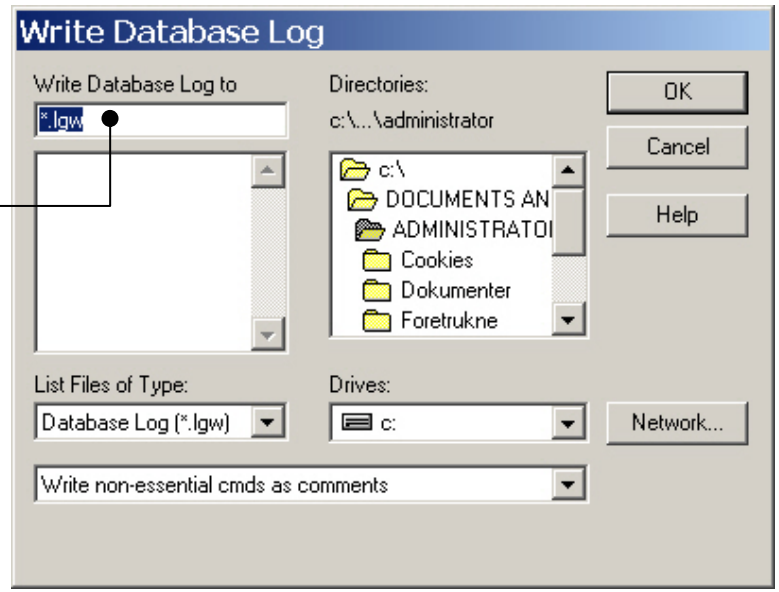
NB: It is often necessary to “Clear” the model for example if Element Type or model geometry is to be changed

Select all areas defined to be meshed

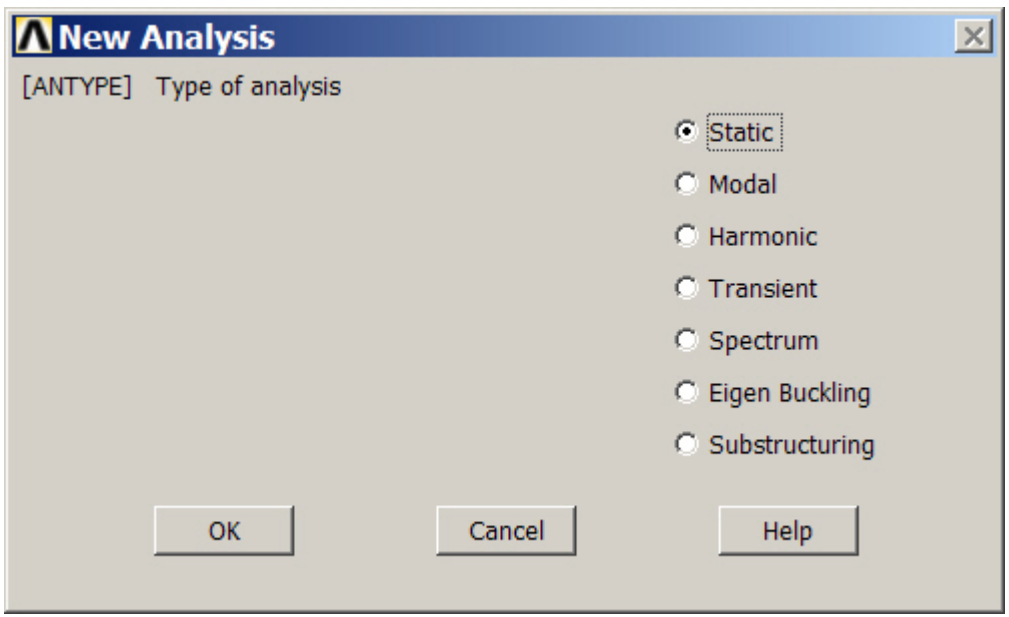
Example – Analysis Type

File > Write DB log file

Enter “example0600.lgw”

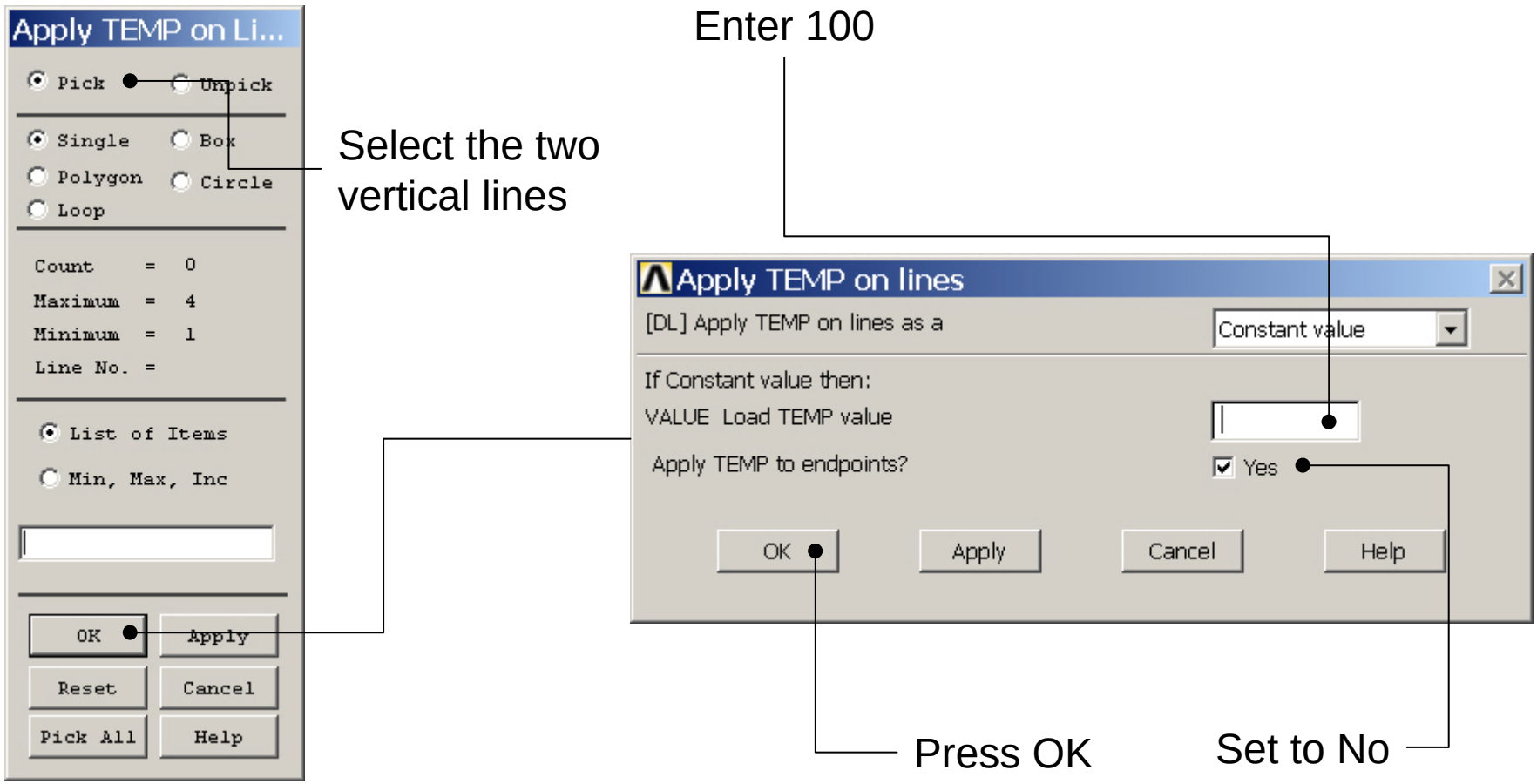


Solution > Analysis Type > New Analysis



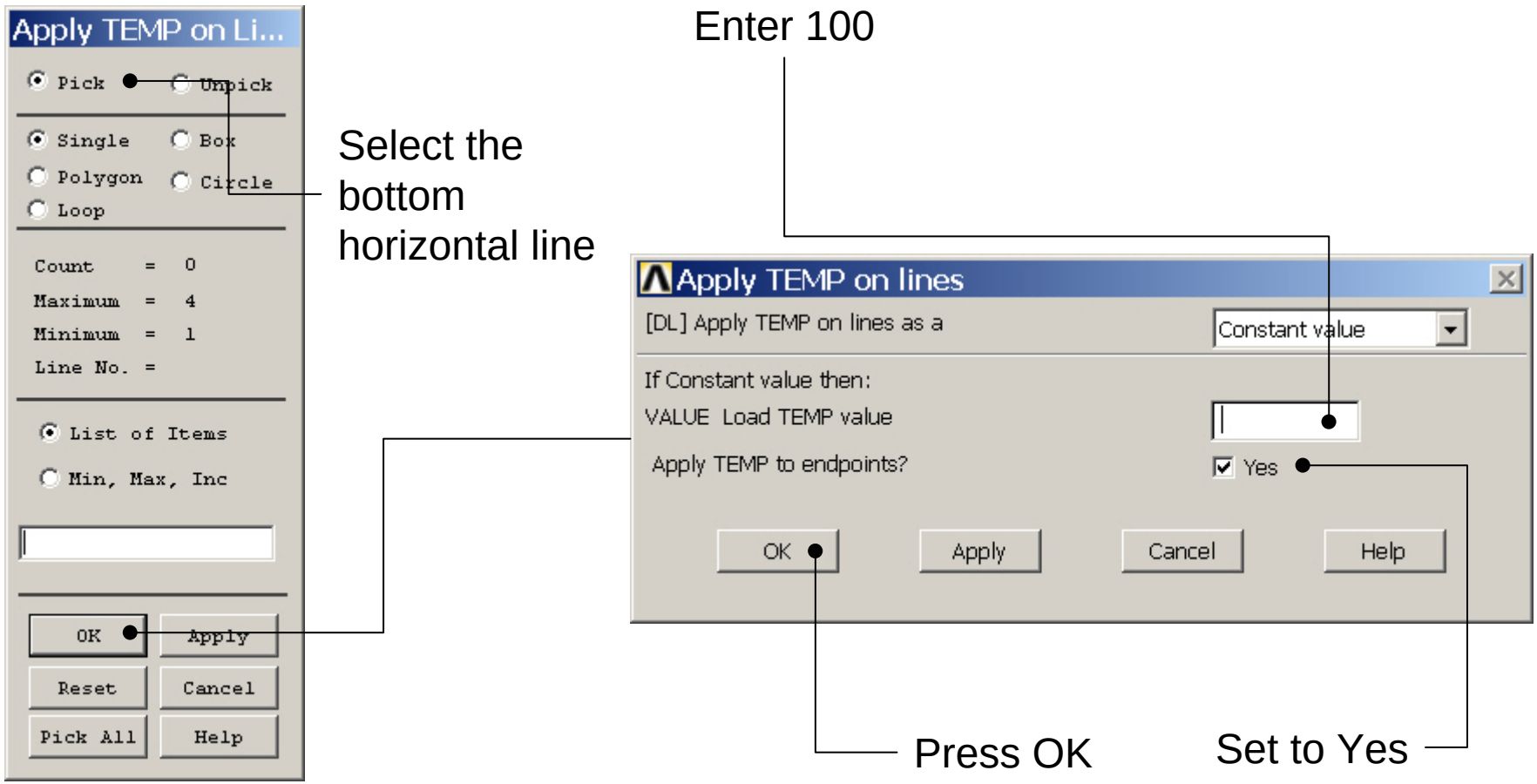
Example – Define Loads

Solution > Define Loads > Apply > Thermal > Temperature > On Lines



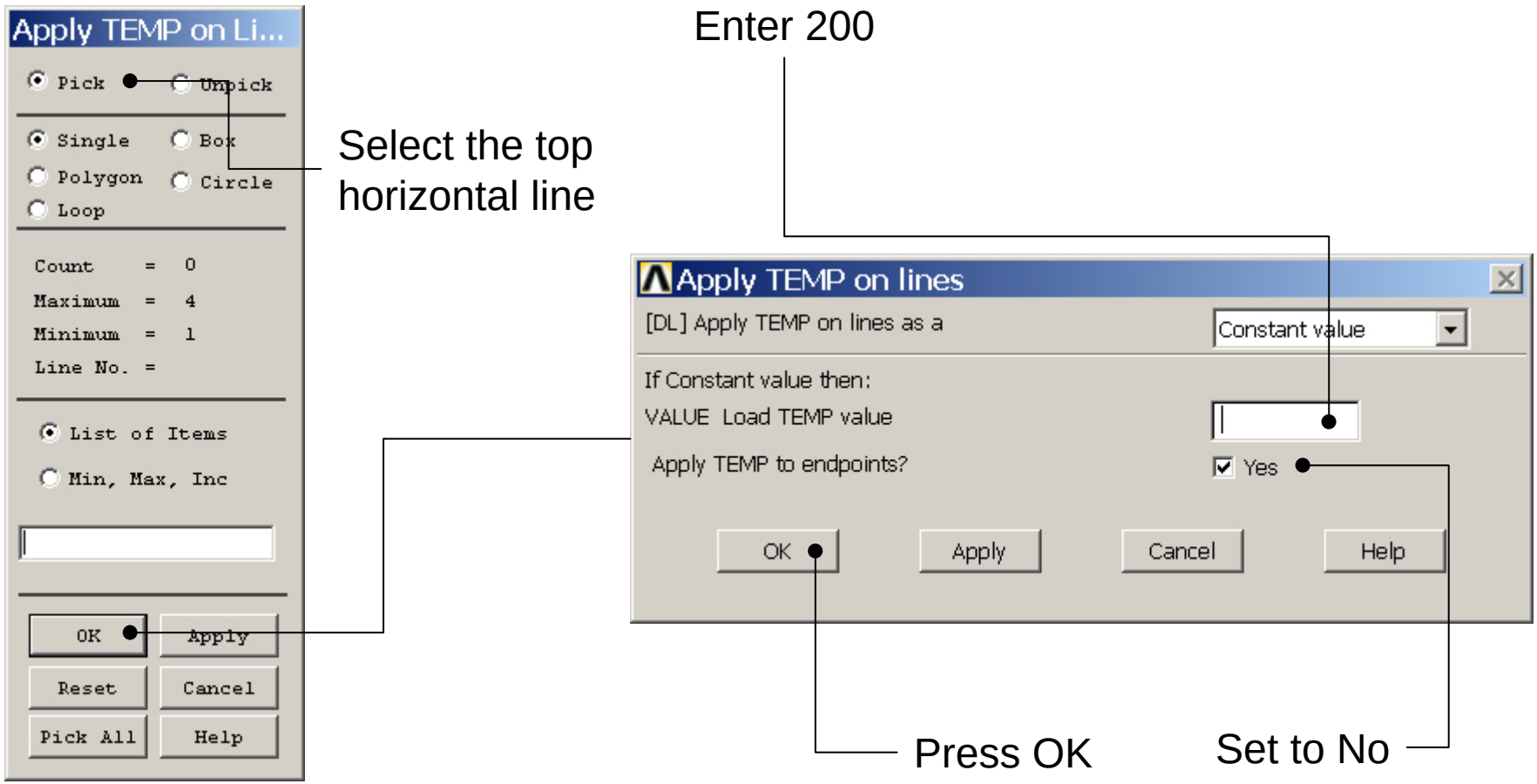
Example – Define Loads

Solution > Define Loads > Apply > Thermal > Temperature > On Lines



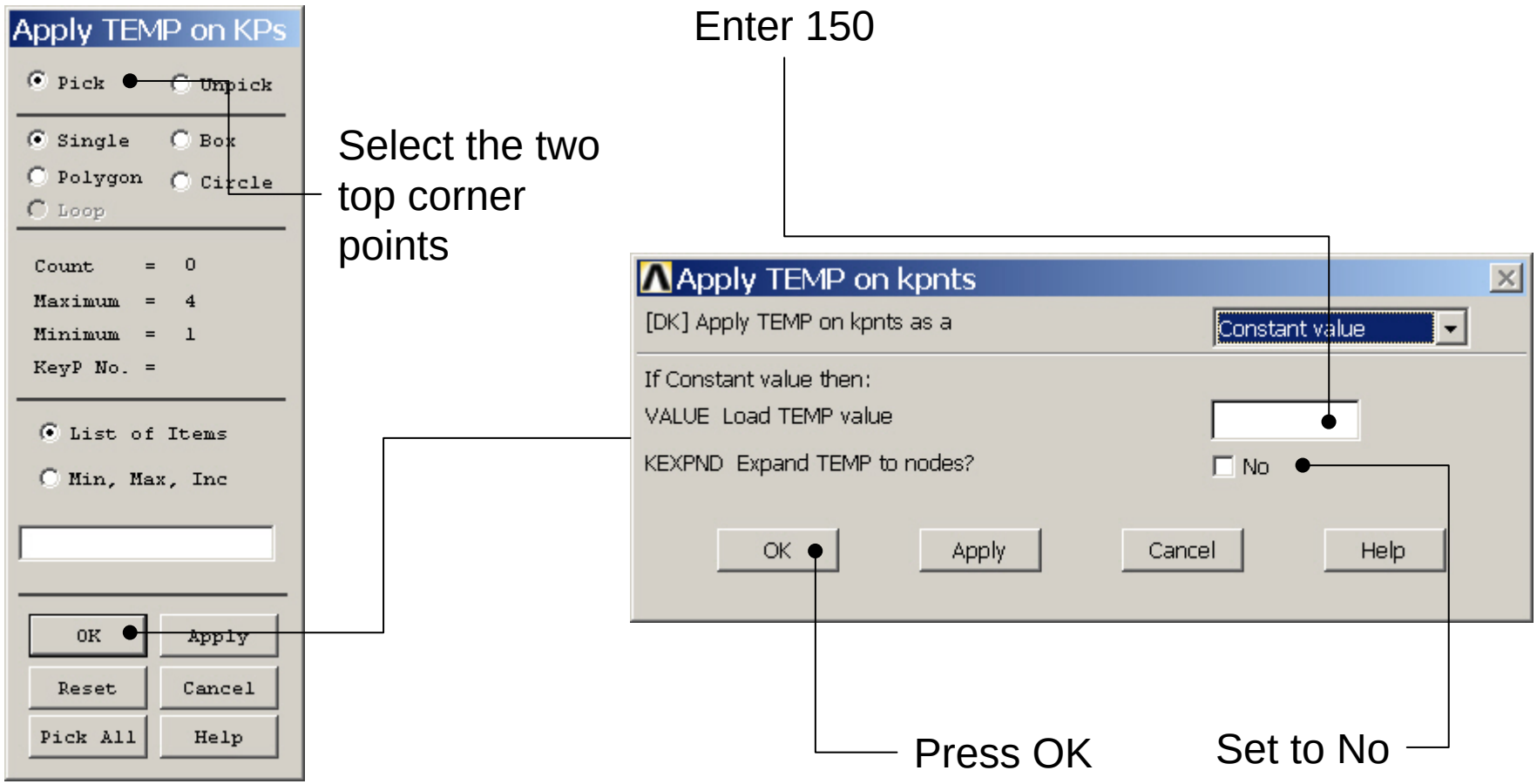
Example – Define Loads

Solution > Define Loads > Apply > Thermal > Temperature > On Lines

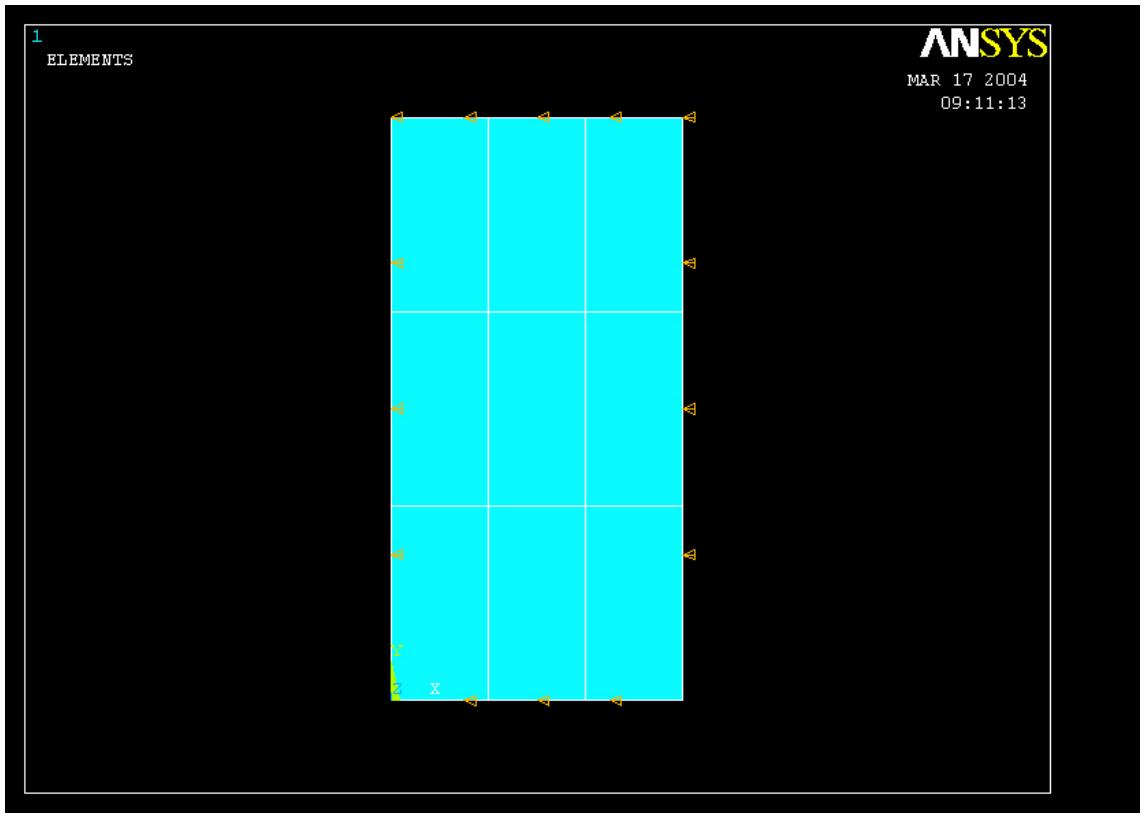


Example – Define Loads

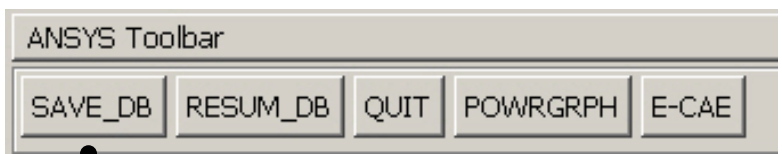
Solution > Define Loads > Apply > Thermal > Temperature > On Lines



Example - Save



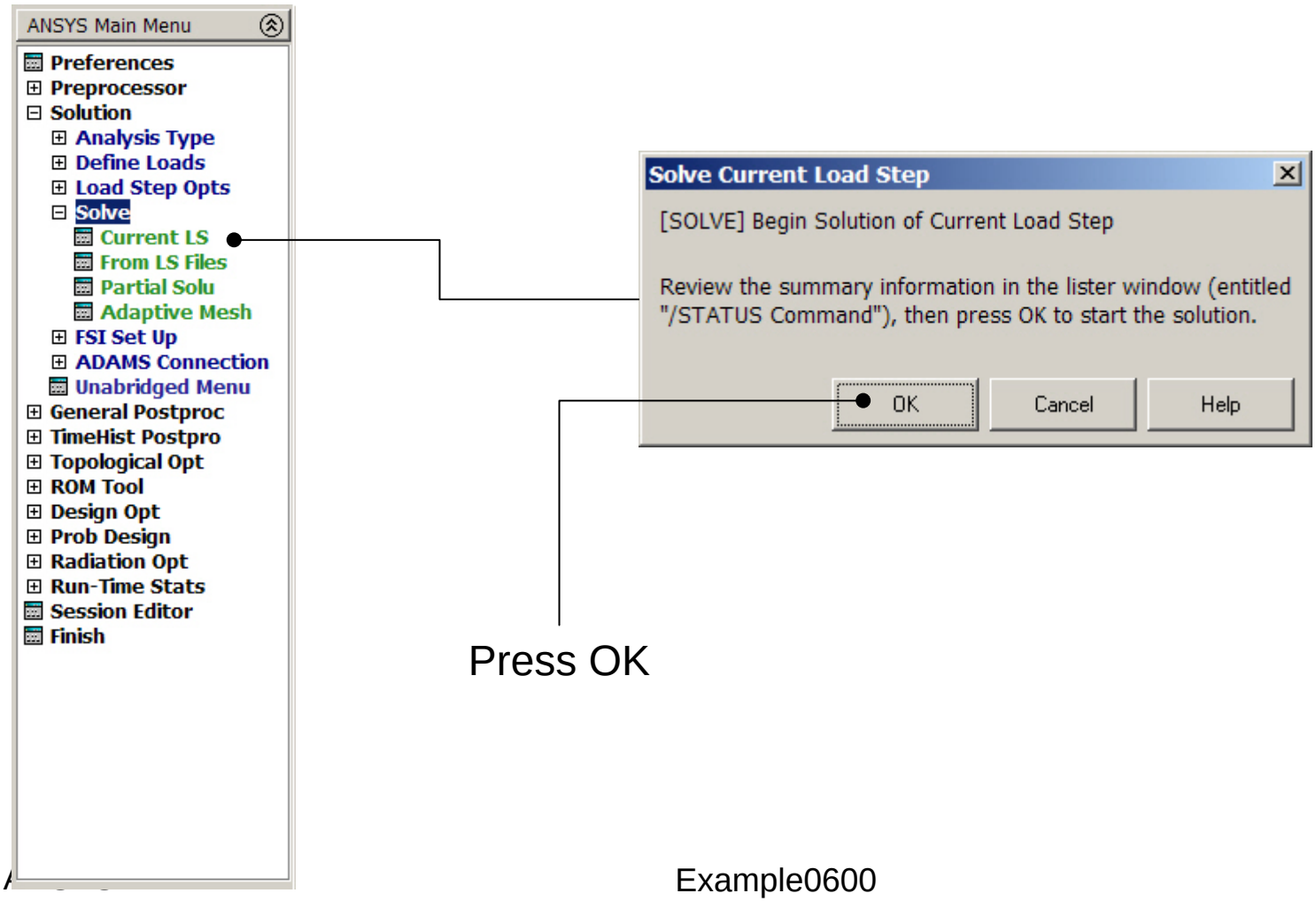
Display of Analysis model



Save the model

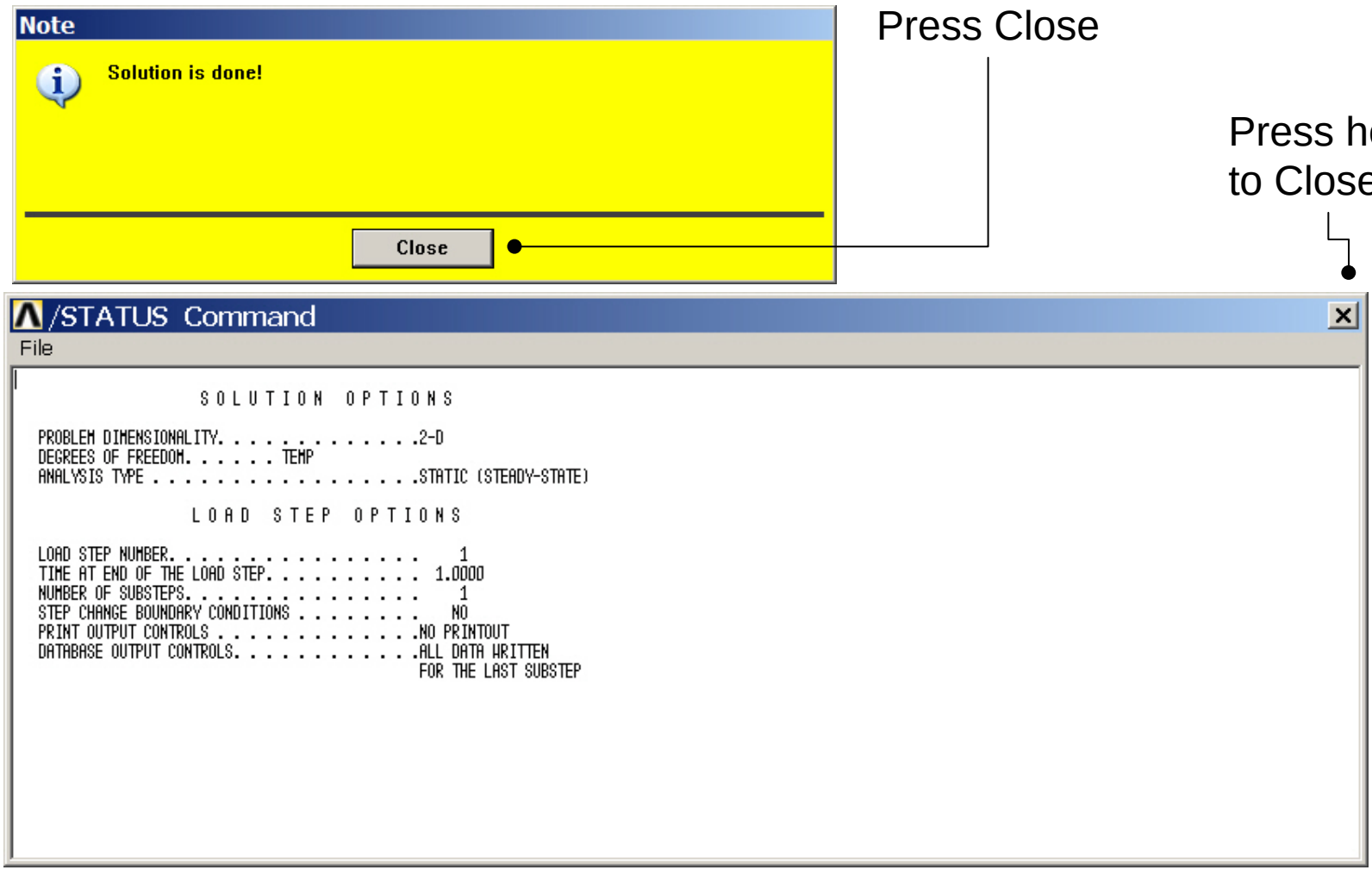
Example - Solve

Solution > Solve > Current LS



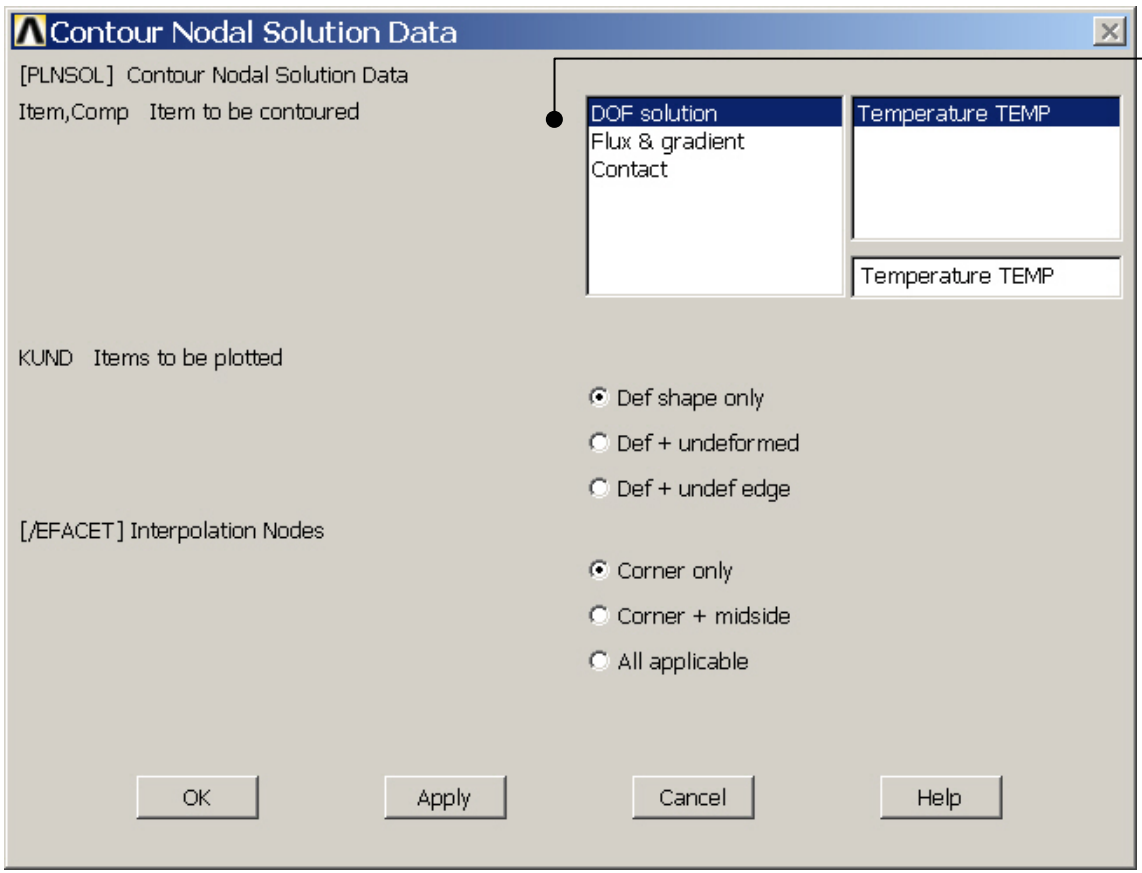
Press OK

Example - Solve



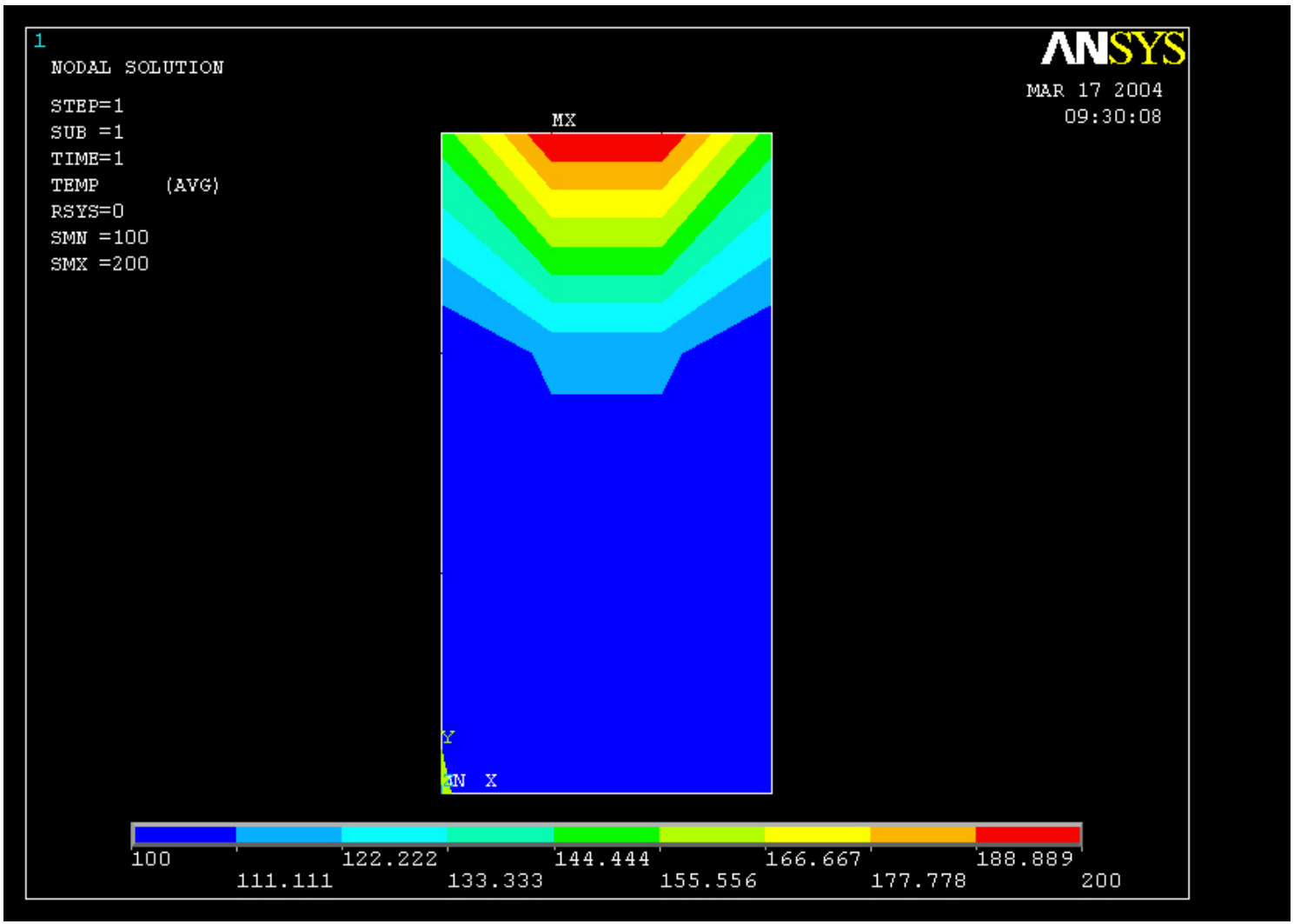
Example – Contour Plot

General Postproc > Plot Results > Contour Plot > Nodal Sol

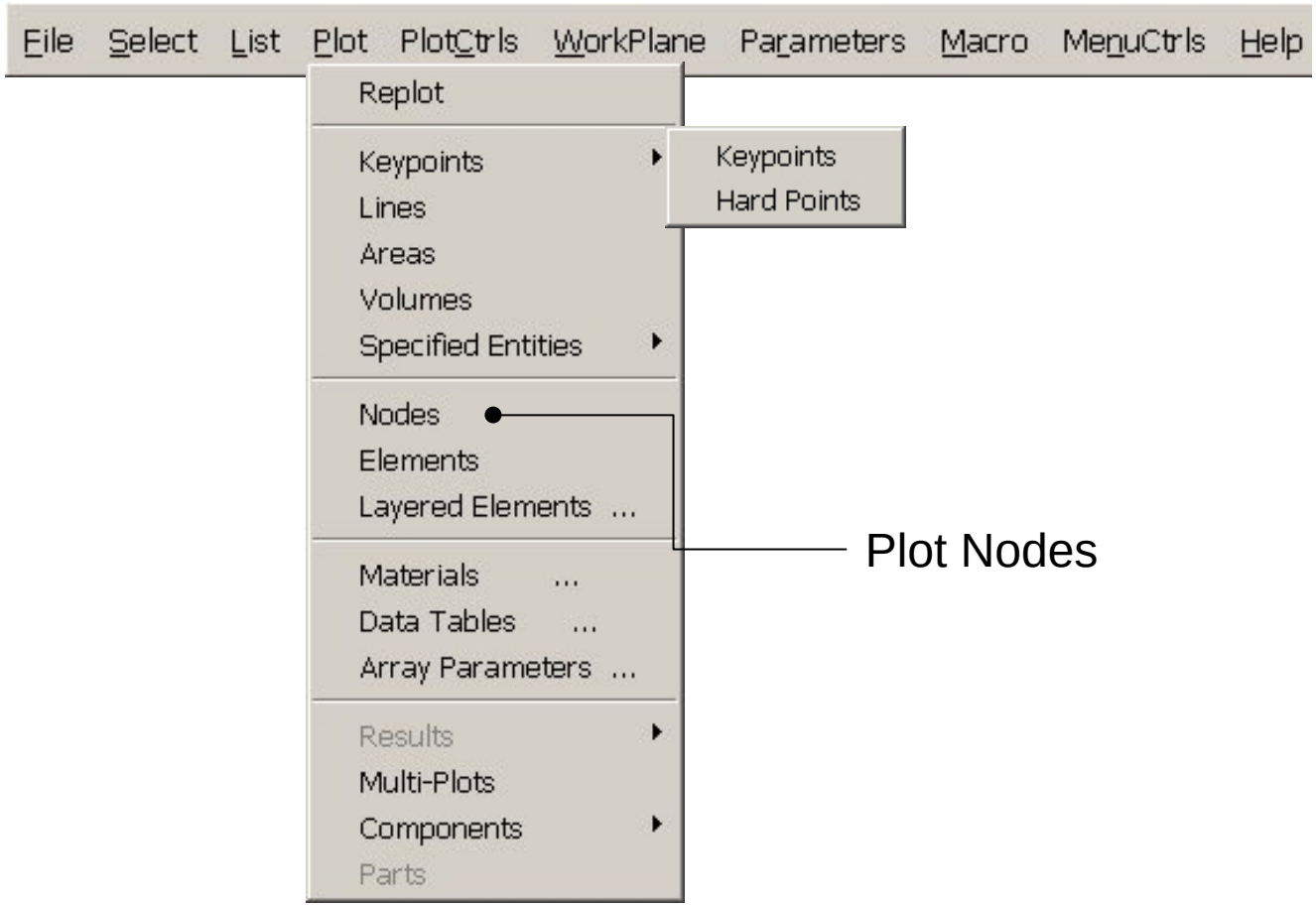


Select DOF solution

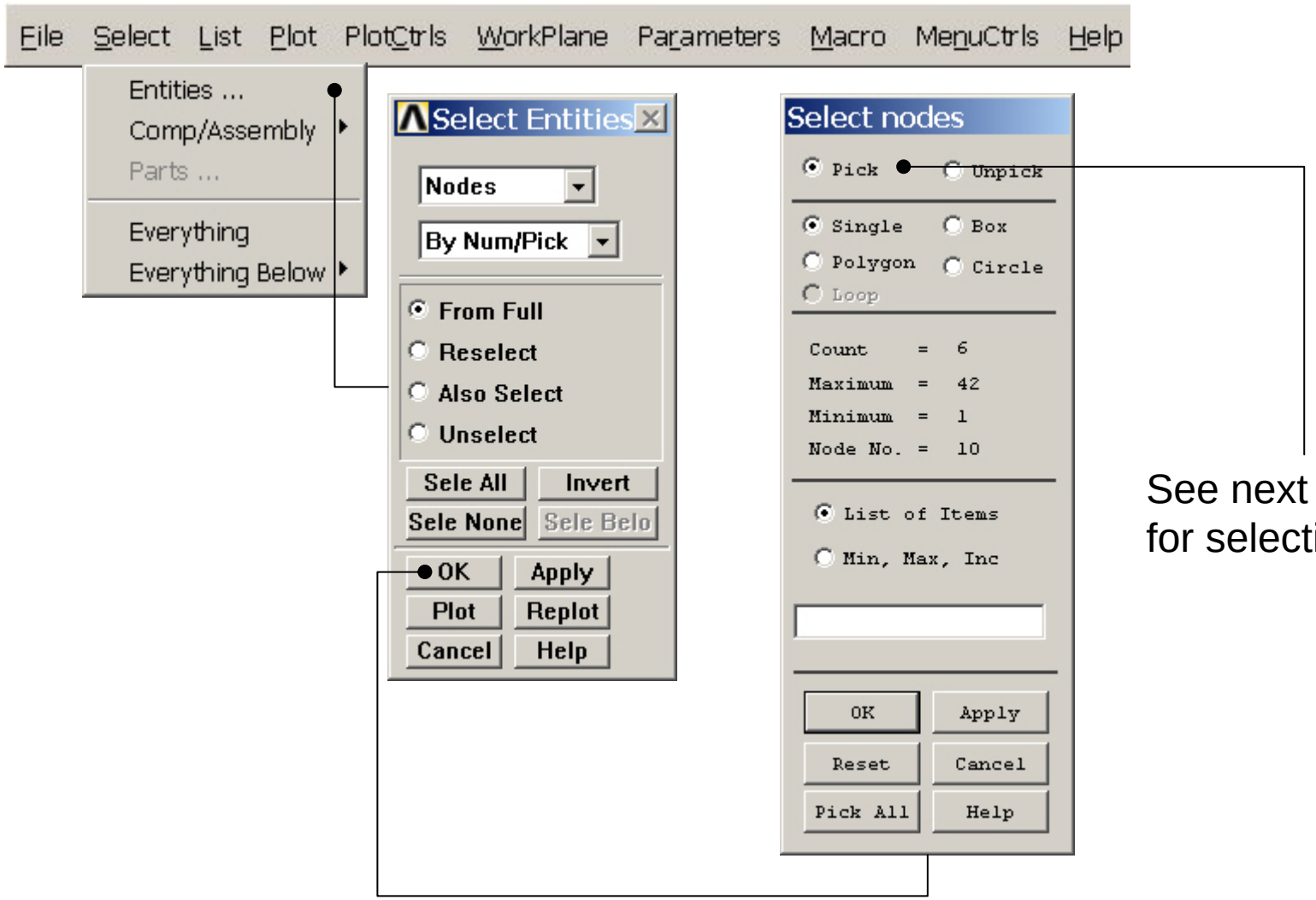
Example – Contour Plot



Example - Plot - Nodes

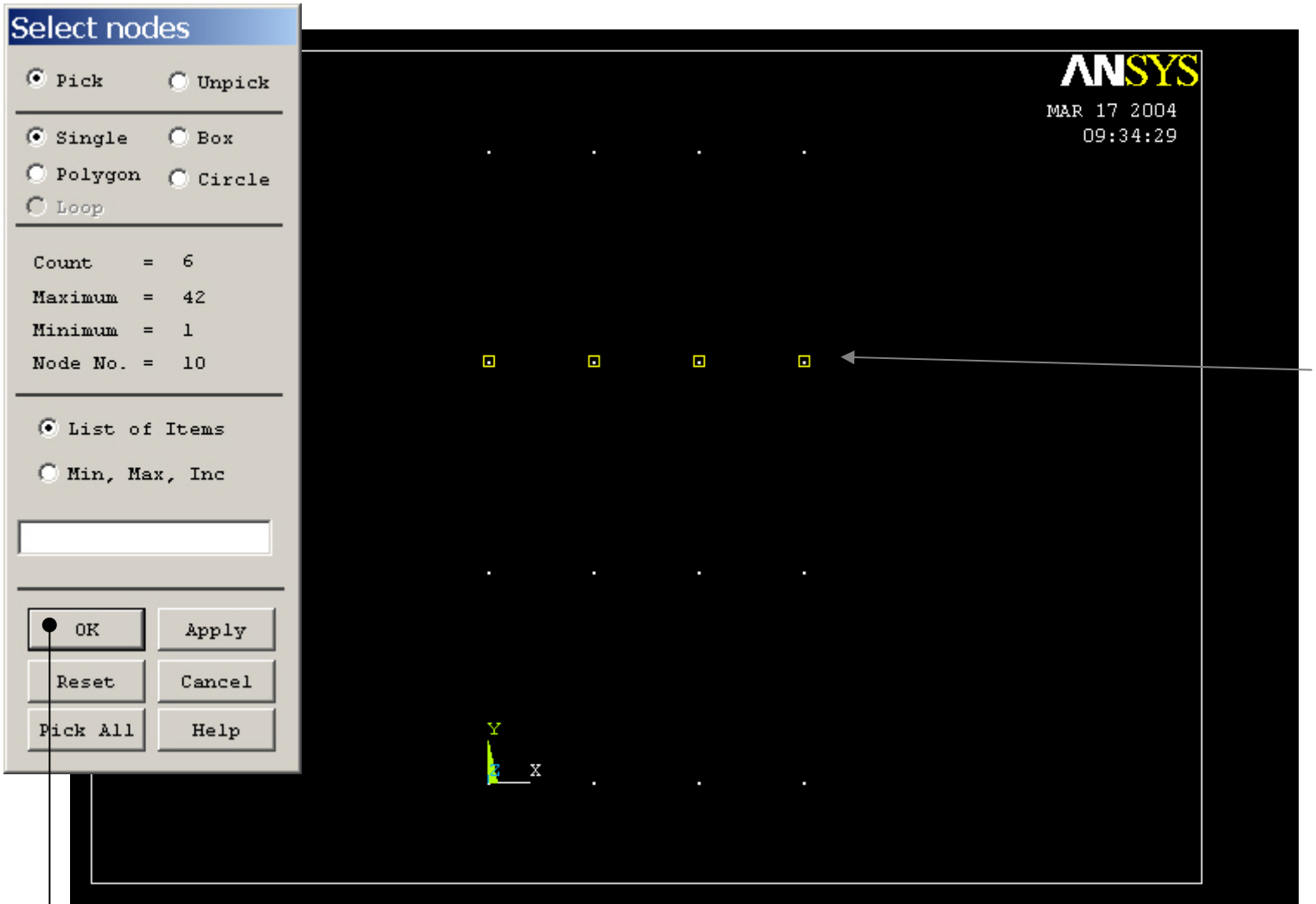


Example – Select - Entities



See next page for selection

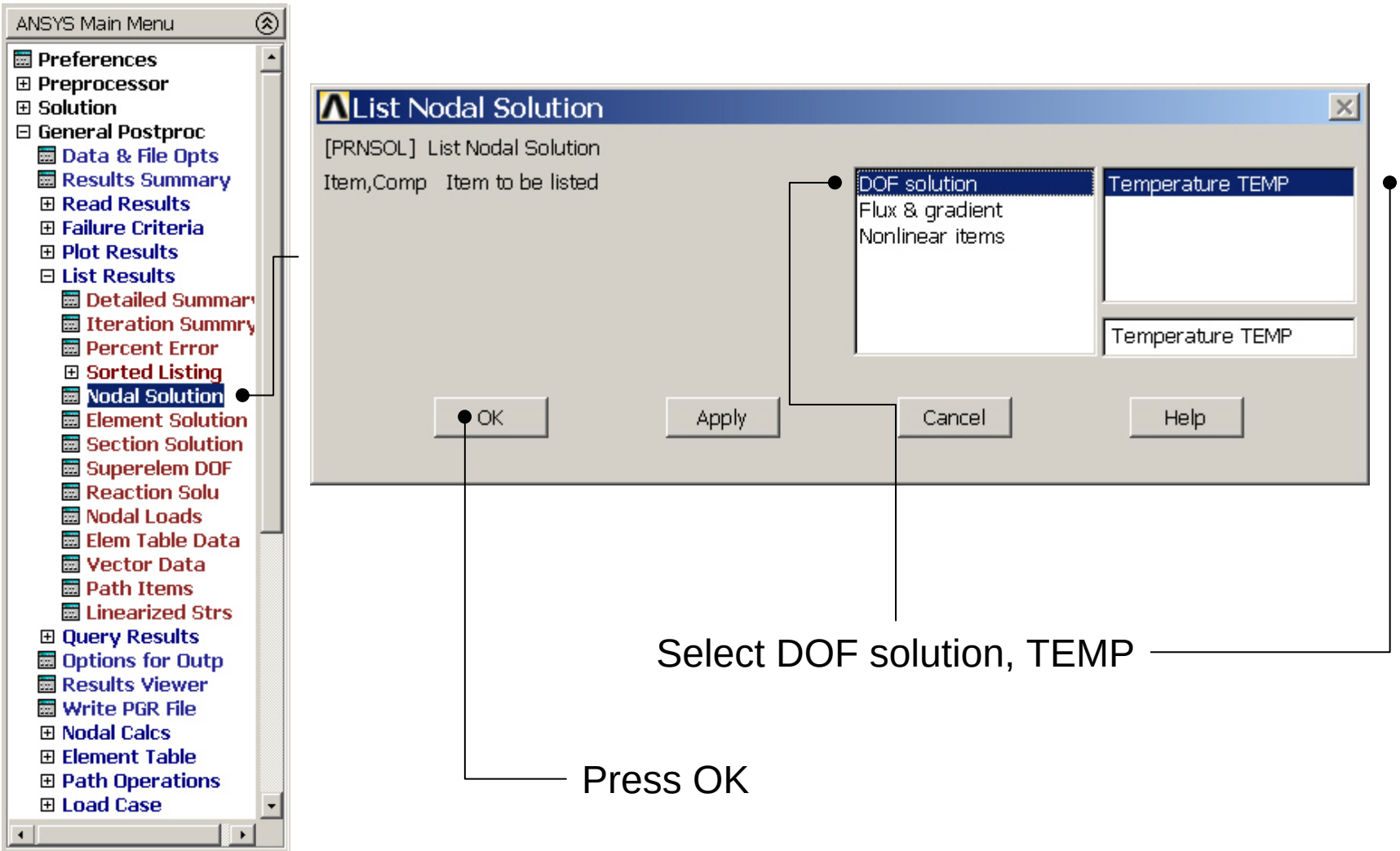
Example – Select Nodes



Select the indicated nodes

Enter OK when finished

Example – List Results



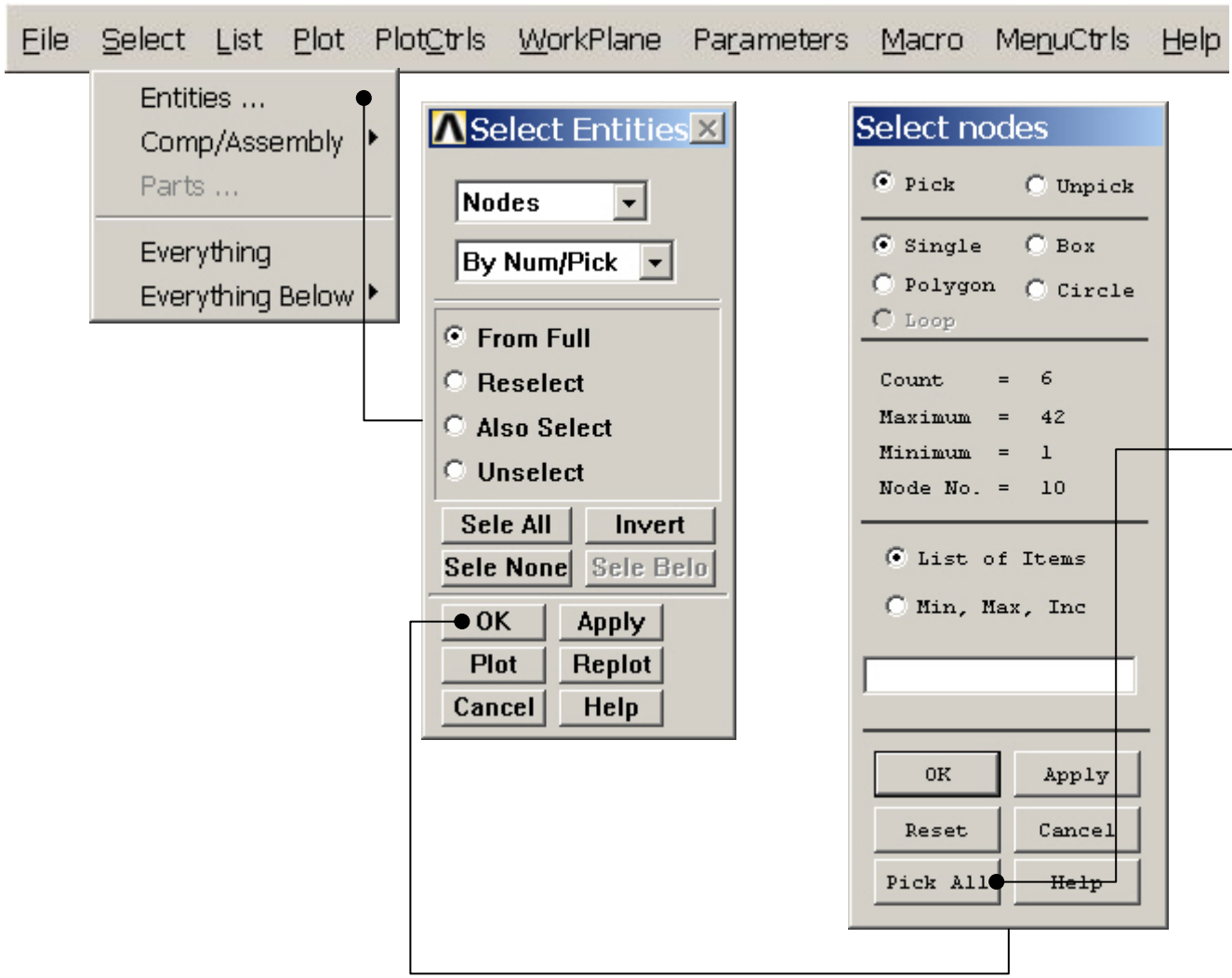
Example – List Results

PRNSOL Command

File

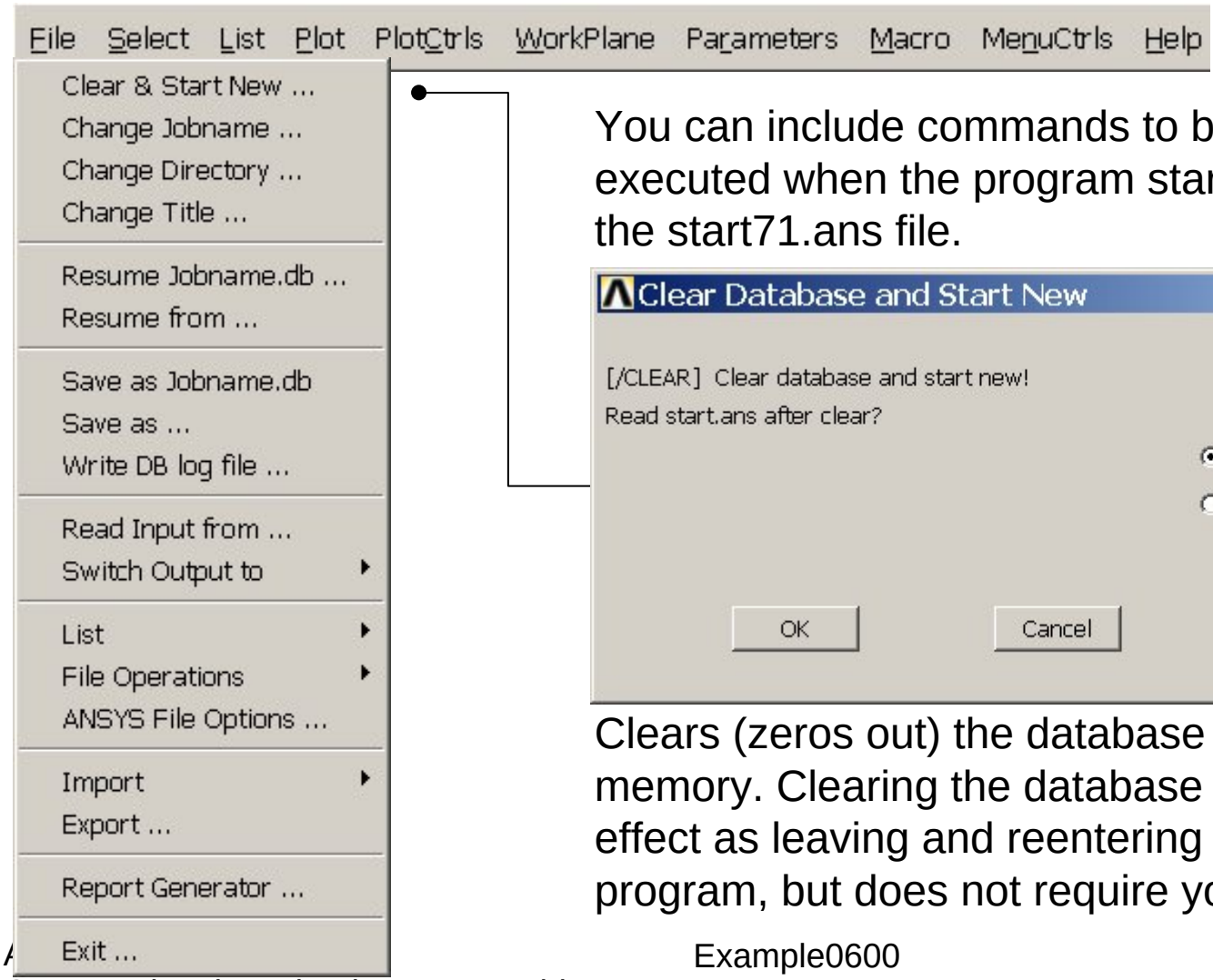
```
PRINT TEMP NODAL SOLUTION PER NODE

 ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****

 LOAD STEP=     1  SUBSTEP=     1
  TIME=    1.0000      LOAD CASE=   0

   NODE      TEMP
      7    100.00
     11    100.00
     14    113.48
     16    113.48

MAXIMUM ABSOLUTE VALUES
NODE         14
VALUE    113.48
```

Example – Select - Entities

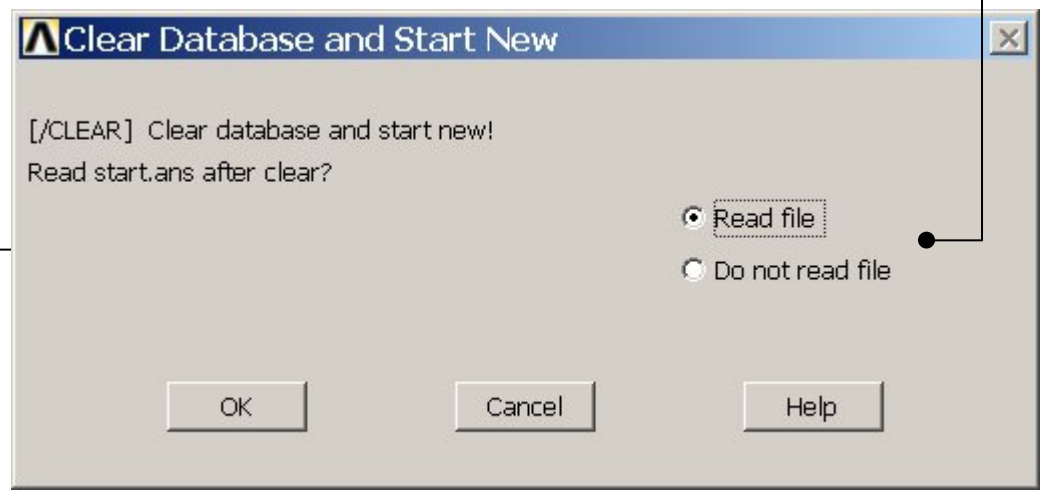


Select Pick All

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.