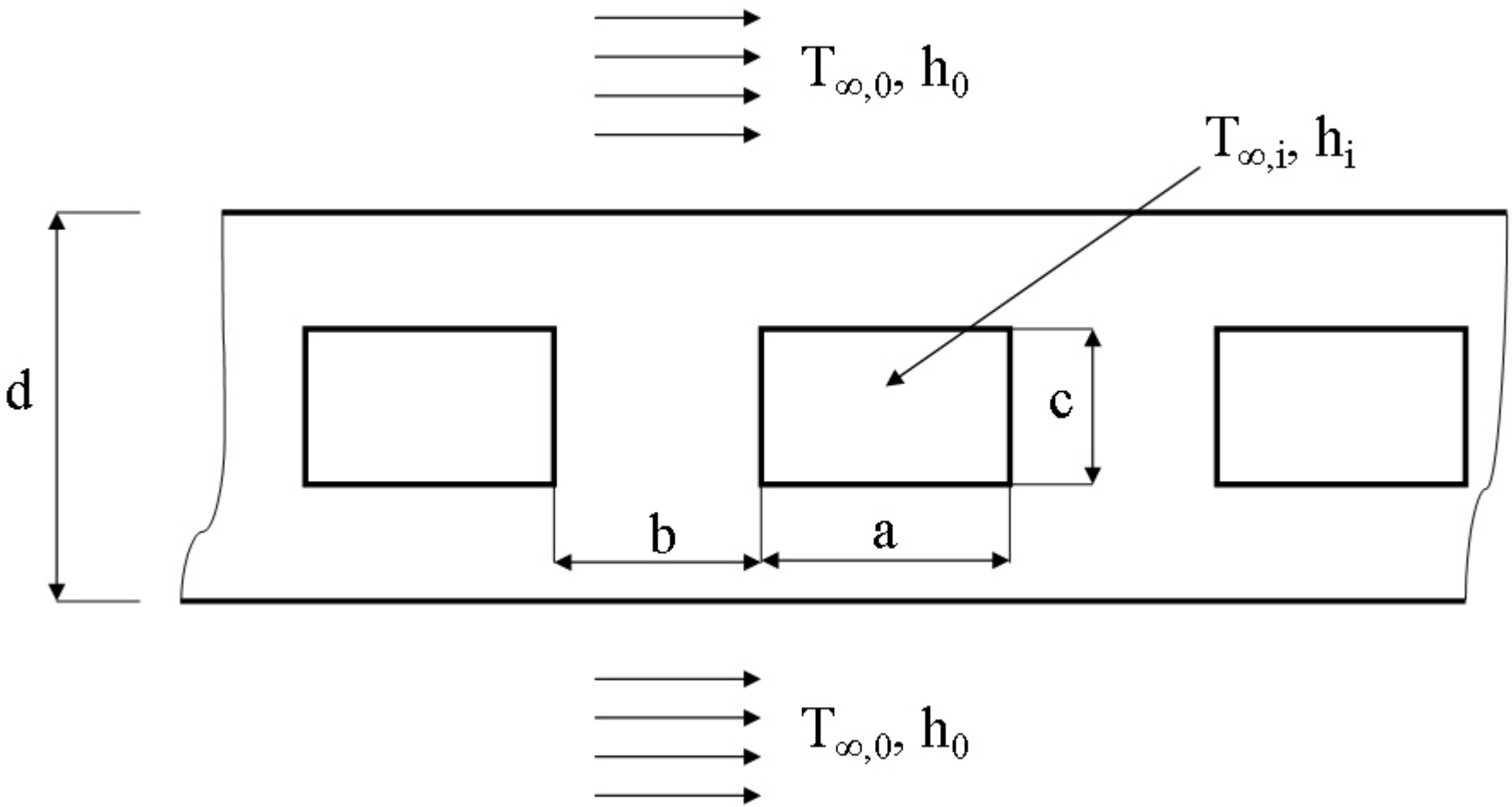
Course in ANSYS

Example0601

Example – Turbine Blade



Example – Turbine Blade

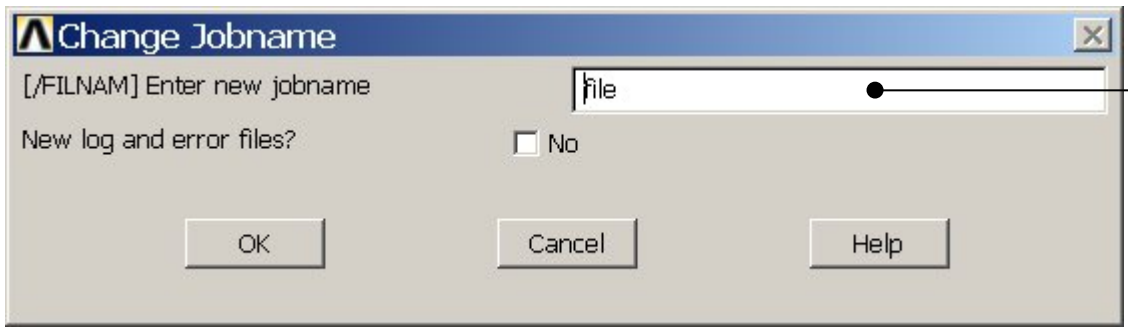
- **Objective:** Solve for the temperature distribution within the 6mm thick turbine blade, with 2mm x 6mm rectangular cooling channels.
- **Tasks:** How should this be modelled?
- **Topics:** Element type, thermal analysis, modeling, plot results, output graphics

$E = 210000 N/mm^2$
$\nu = 0.3$
Thermal conductivity of blade = 25W/(m-K)
Operating conditions:
$T_{4,o} = 1700K$
$h_o = 1000 W/(m^2\text{-}K)$
$T_{4,i} = 400K$
$h_i = 200 W/(m^2\text{-}K)$
A = 6mm
B = 4mm
C = 2mm
D = 6mm

Example - title

Utility Menu > File > Change Jobname — GUI

/jobname, Example0601 — Command line entry

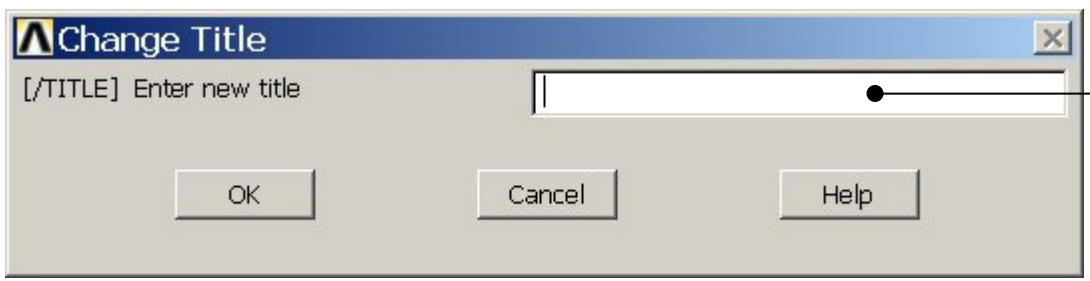


Enter: Example0601

Utility Menu > File > Change Title

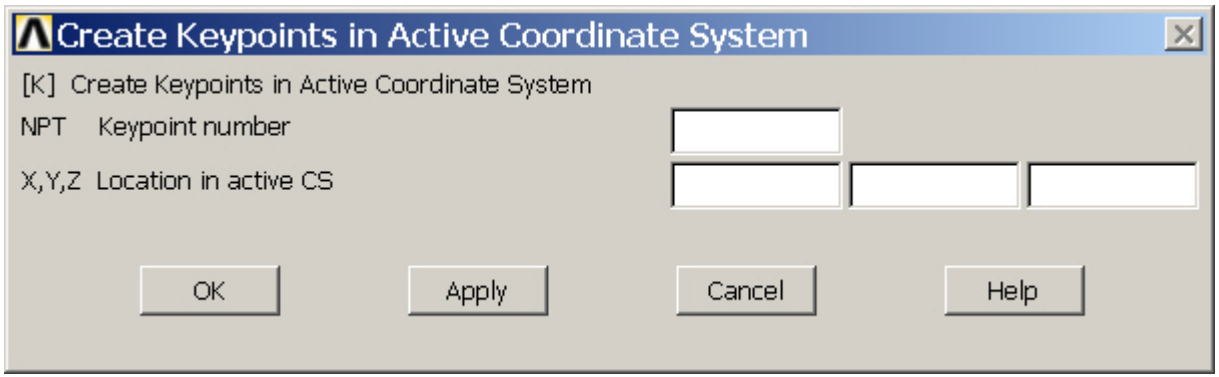
/title, Temperature in a Turbine Blade

Enter: Temperature in a Turbine Blade



Example - Keypoints

Preprocessor > Modeling > Create > Keypoints > In Active CS

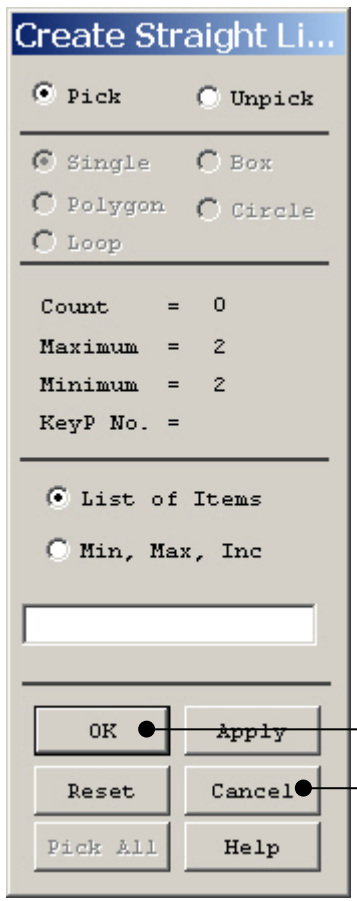


Enter the KP's

Keypoint Number	X-Location	Y-Location
1	-0.005	0.003
2	0	0.003
3	0	0.001
4	-0.003	0.001
5	-0.003	0
6	-0.005	0

Example - Lines

Preprocessor > Modeling > Create > Lines > Lines > Straight Line



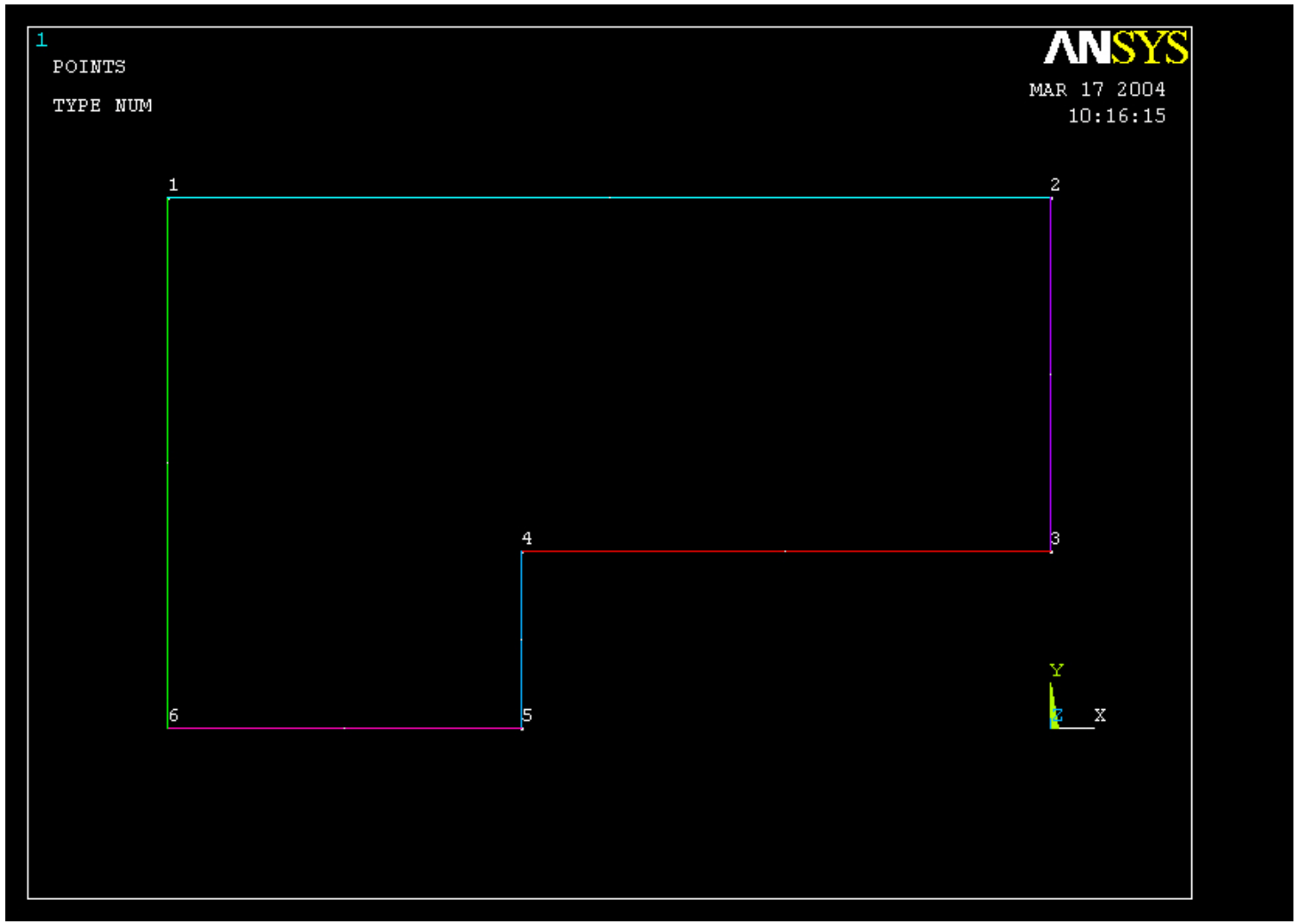
HINT: By clicking with the right-hand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

Pick: upward arrow

Unpick: downward arrow

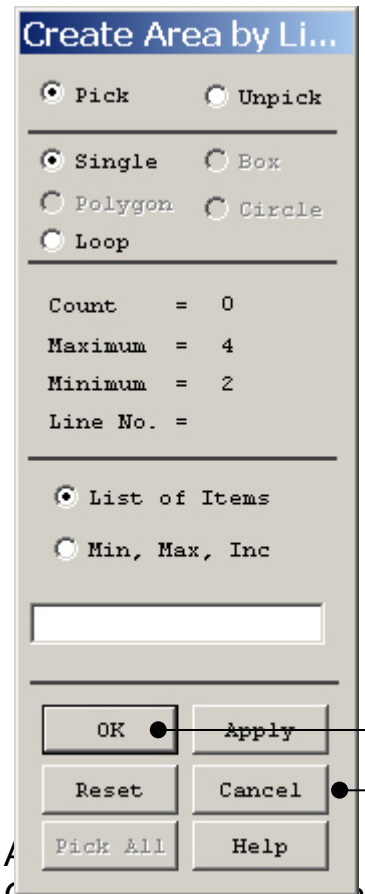
Press OK or Cancel to finish selection

Example - Lines



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



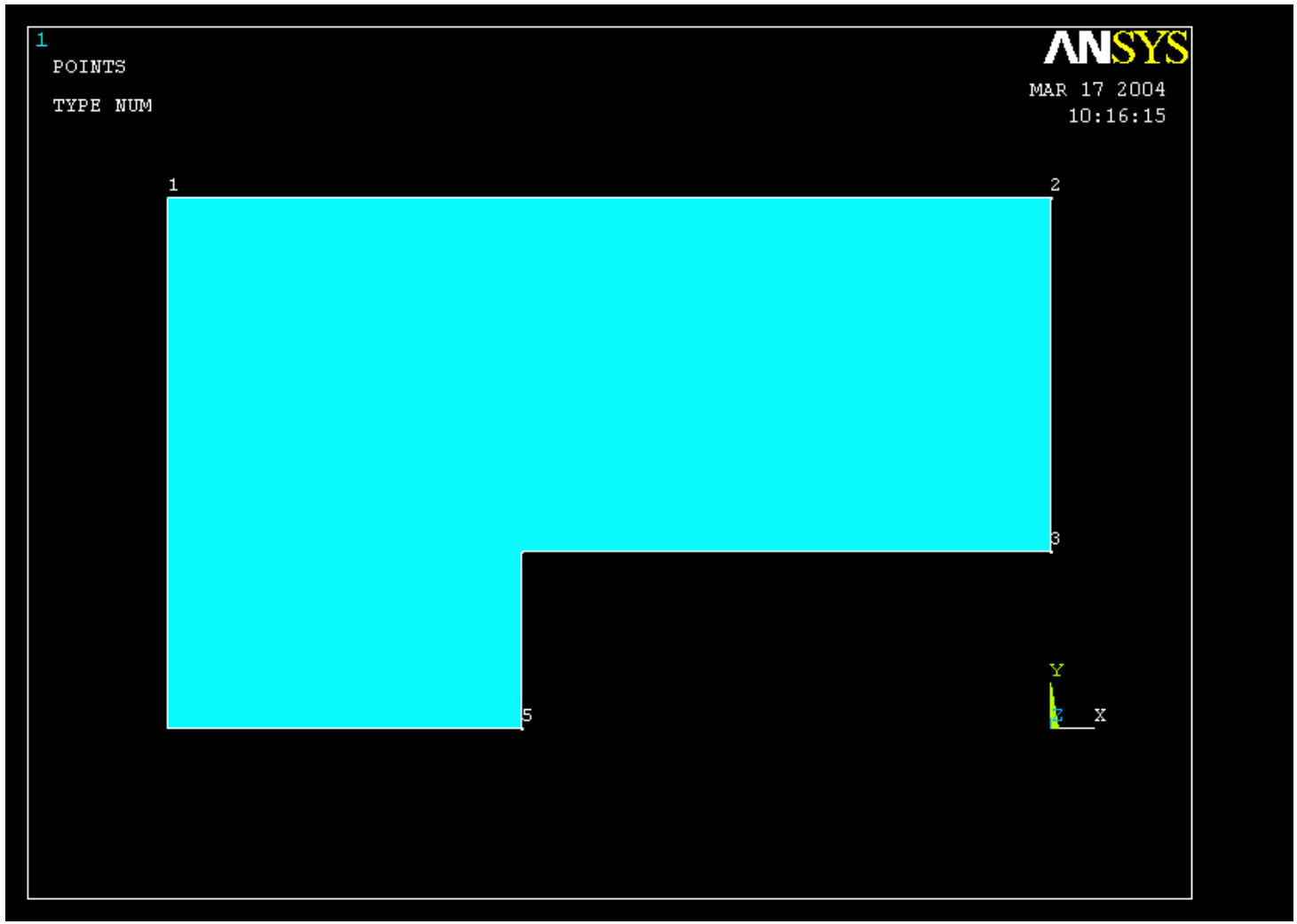
HINT: By clicking with the right-hand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

Pick: upward arrow

Unpick: downward arrow

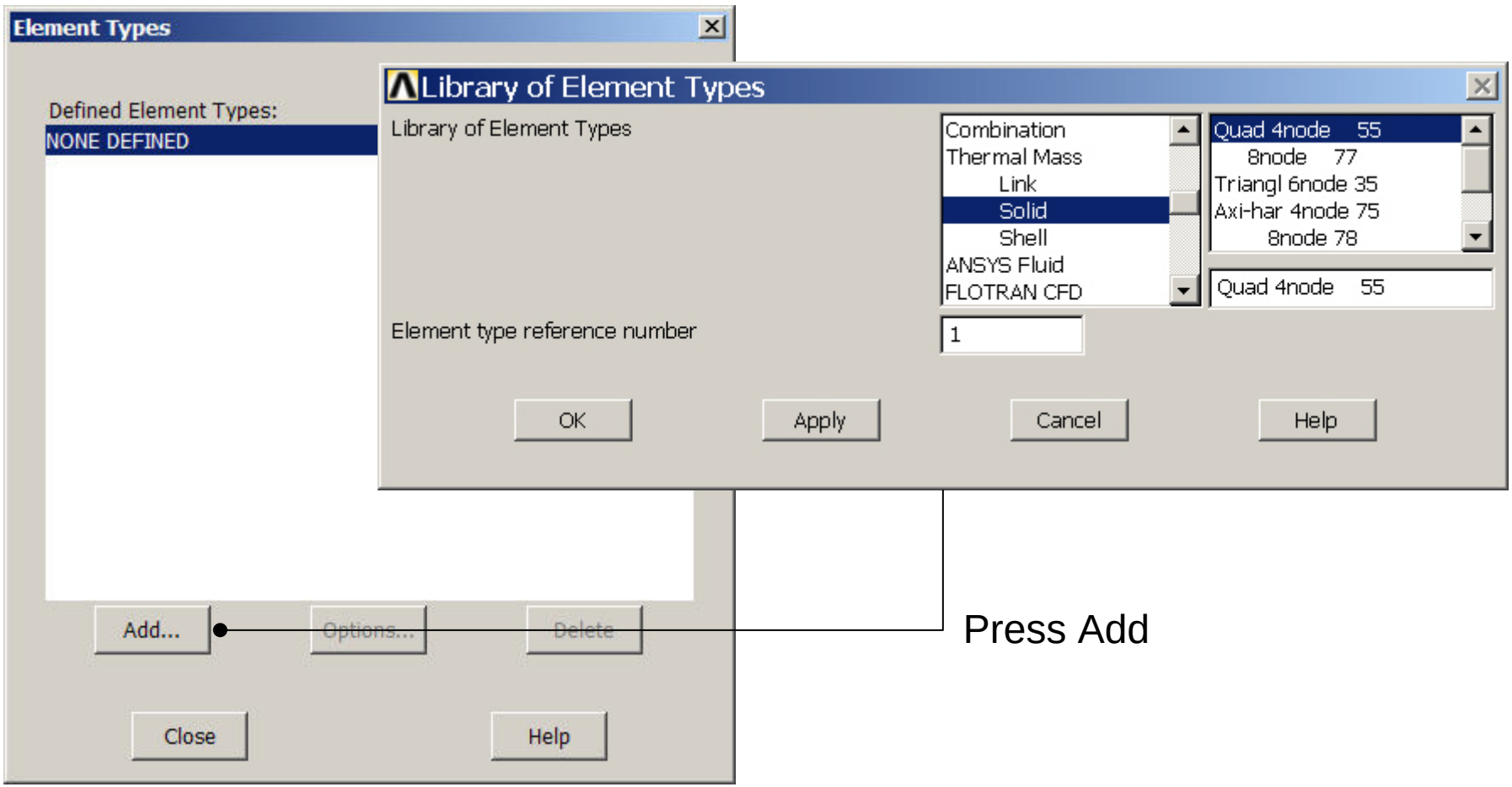
Press OK or Cancel to finish selection

Example – Areas Rectangle



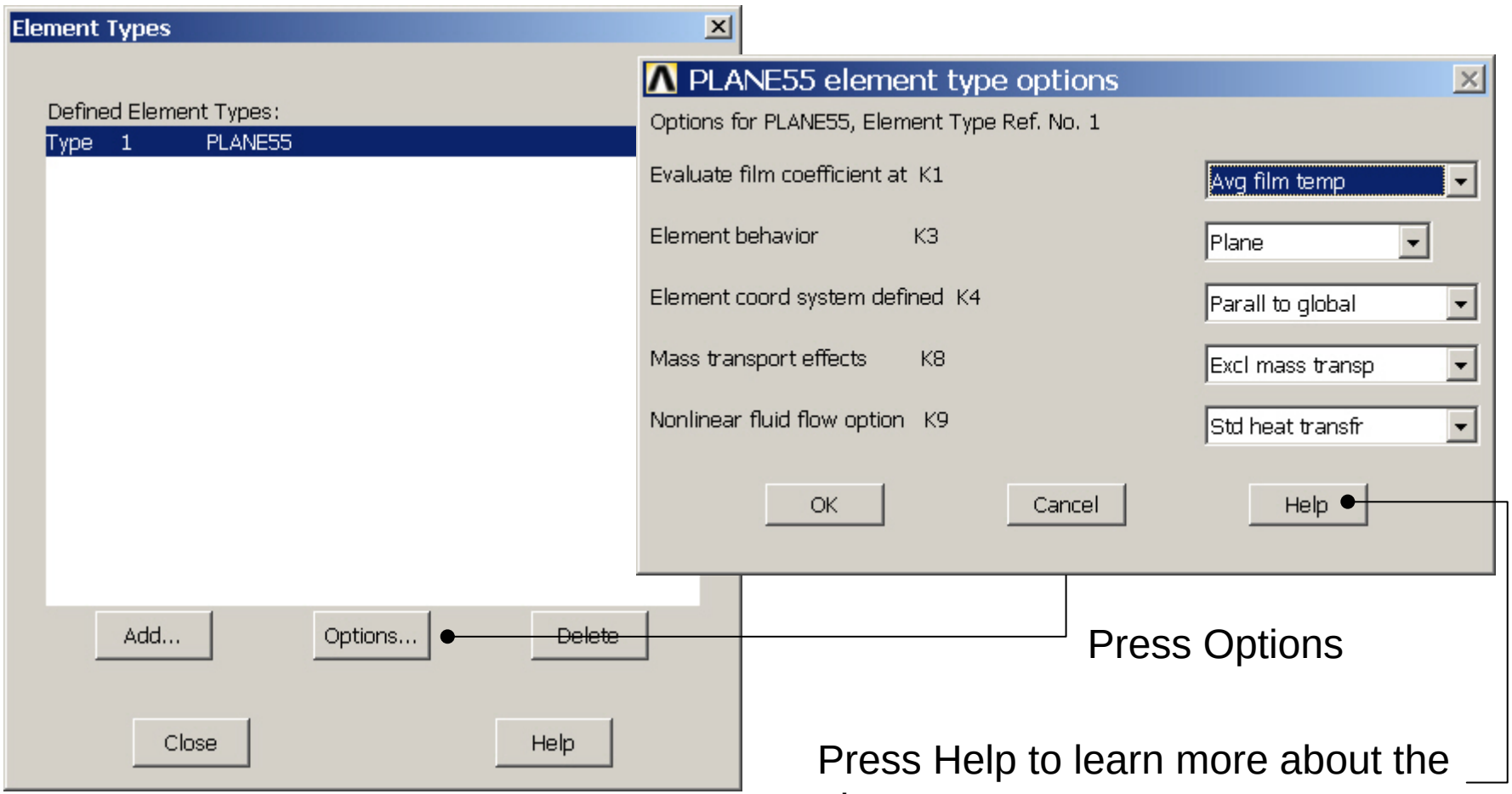
Example – Element Type

Preprocessor > Element Type > Add/Edit/Delete



Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete

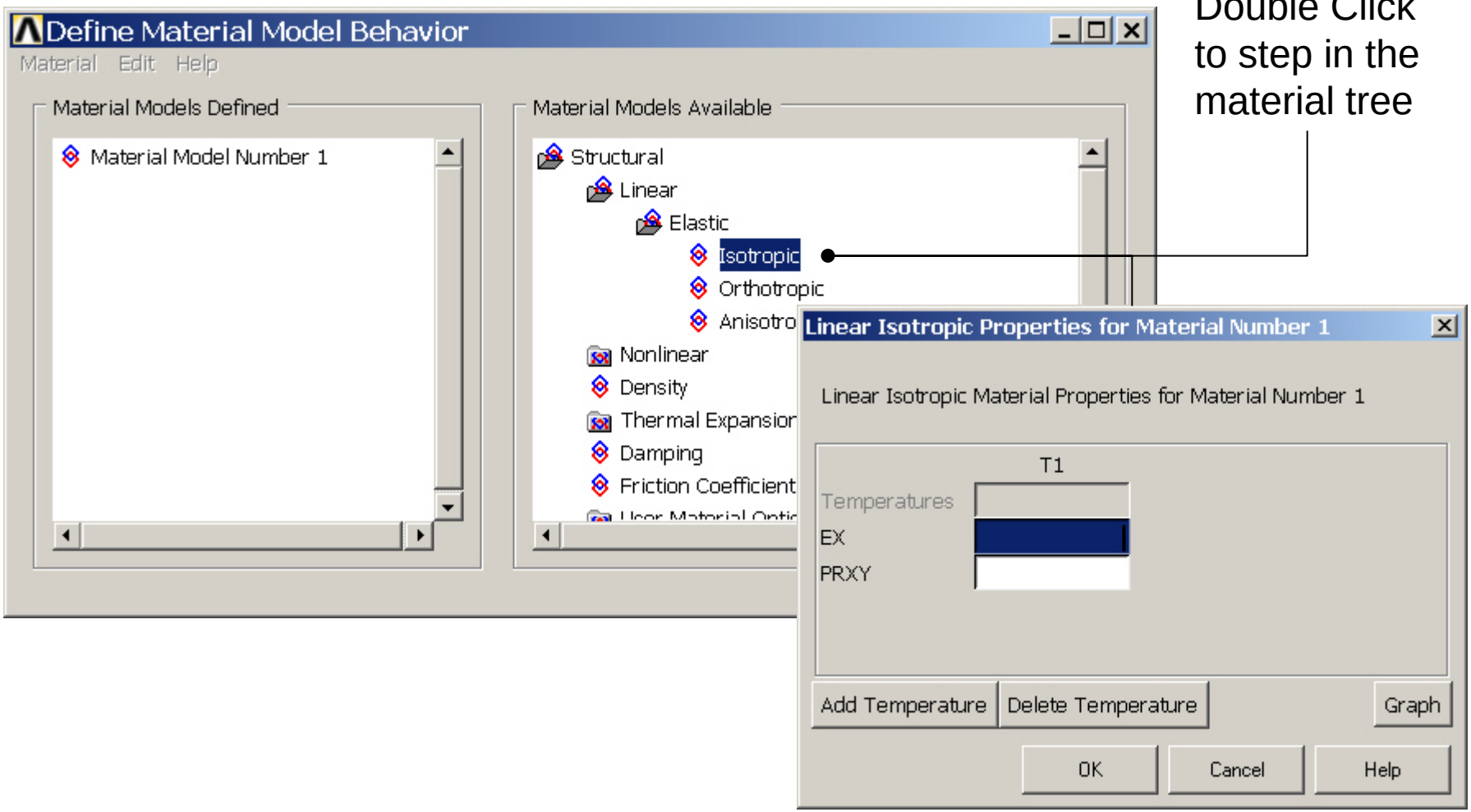


Press Options

Press Help to learn more about the element.

Example - Material Properties

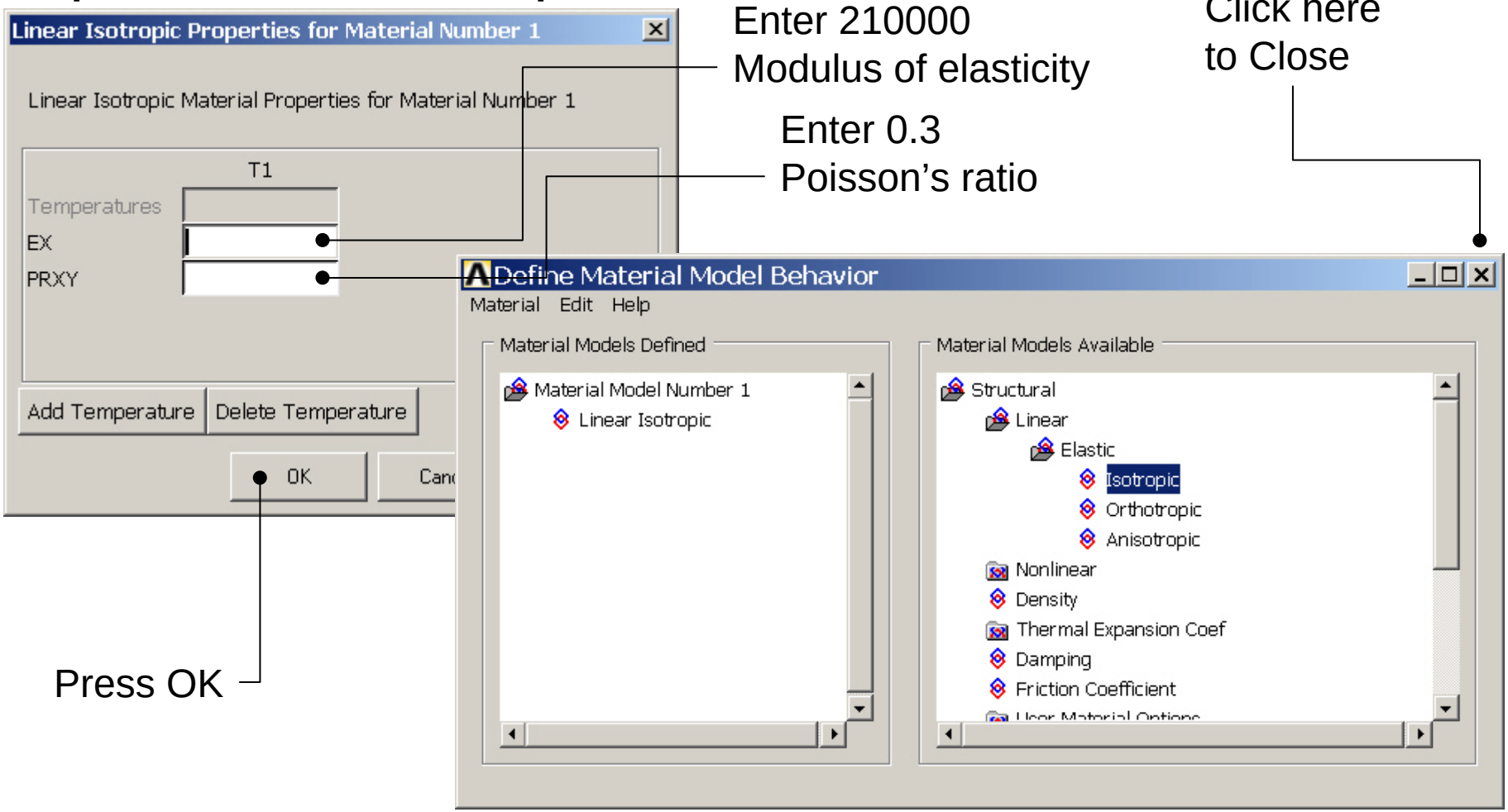
Preprocessor > Material Props > Material Models



Double Click to step in the material tree

Example - Material Properties

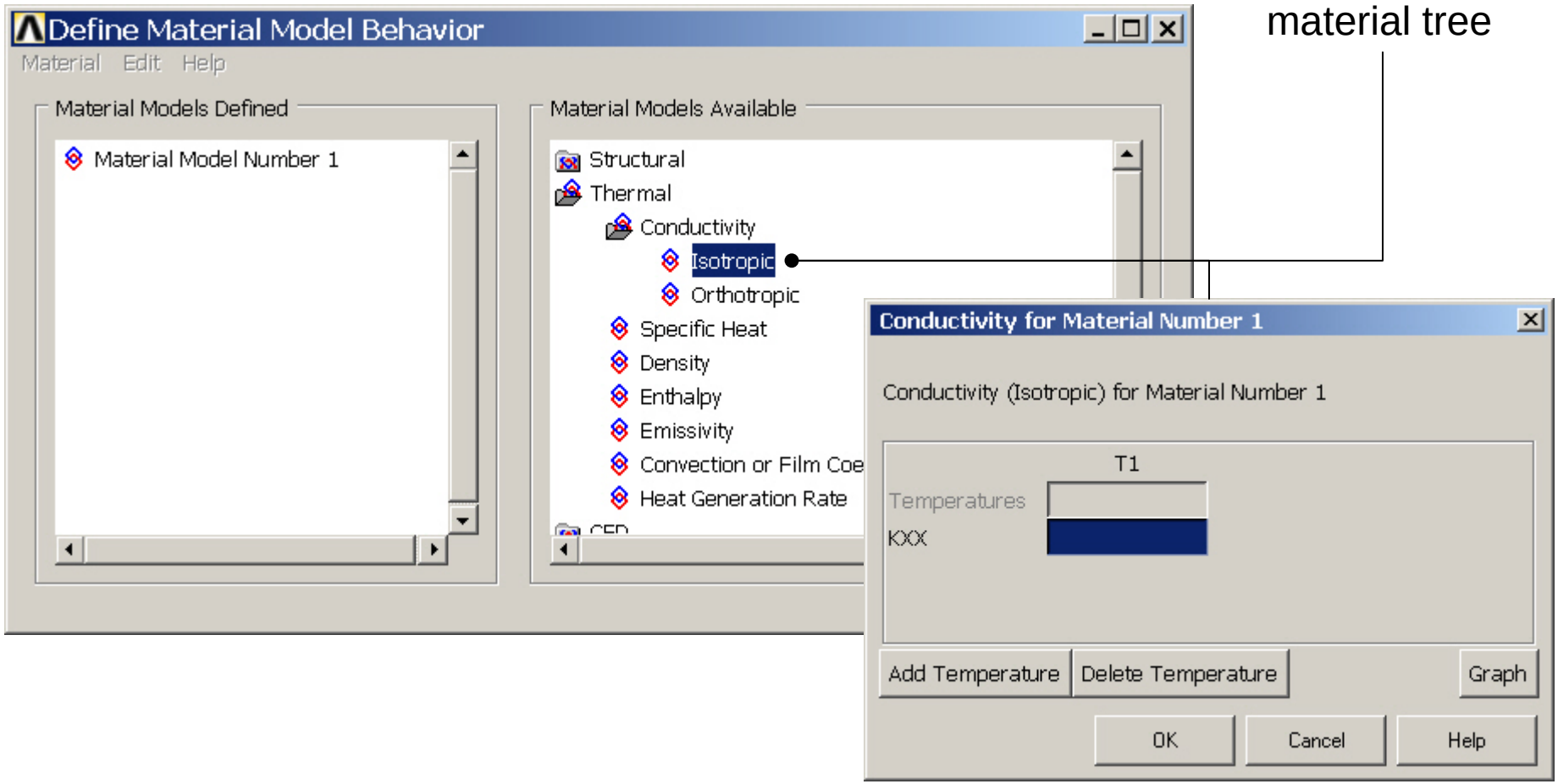
Preprocessor > Material Props > Material Models



Example - Material Properties

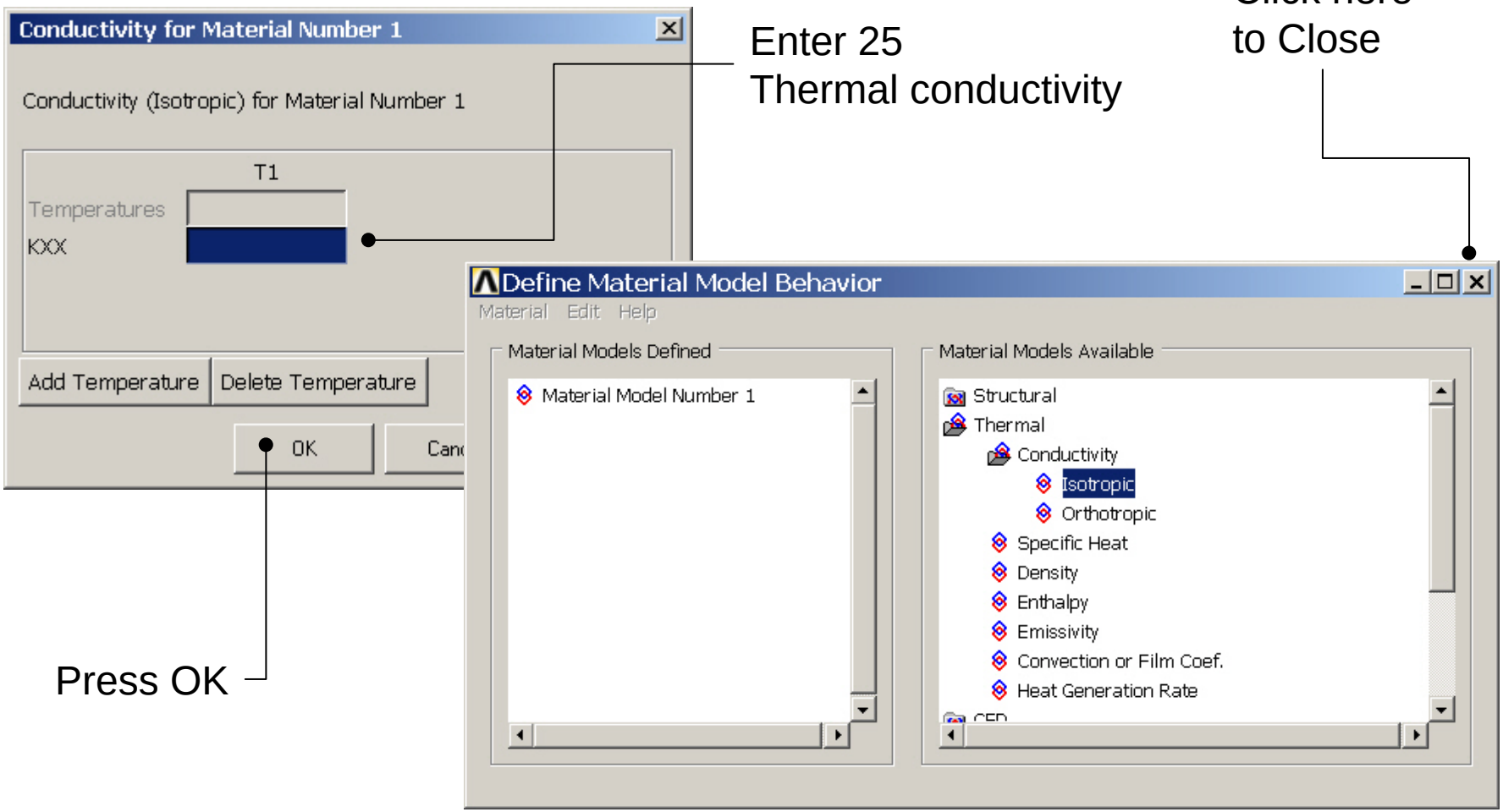
Preprocessor > Material Props > Material Models

Double Click to step in the material tree



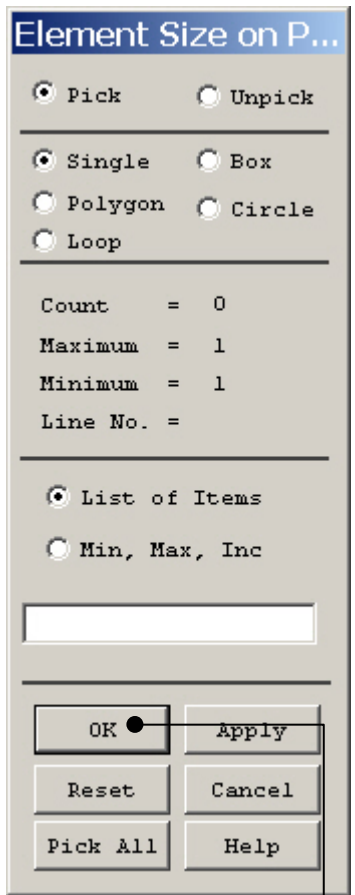
Example - Material Properties

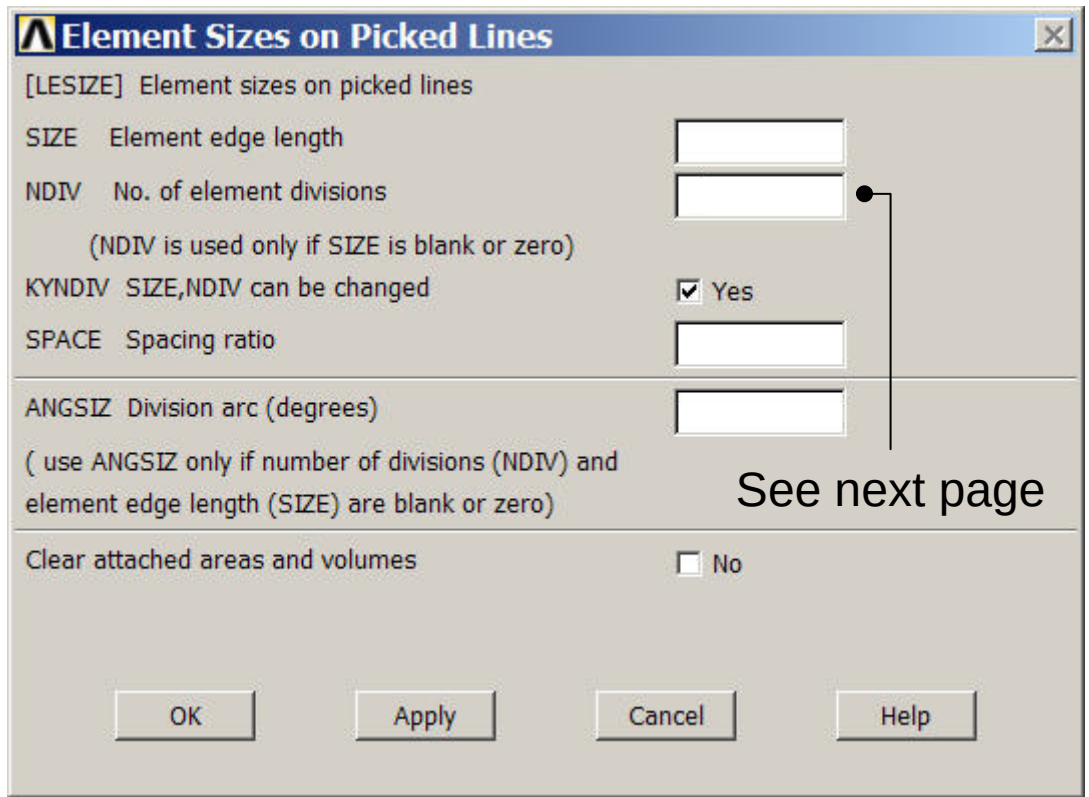
Preprocessor > Material Props > Material Models



Example - Meshing

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





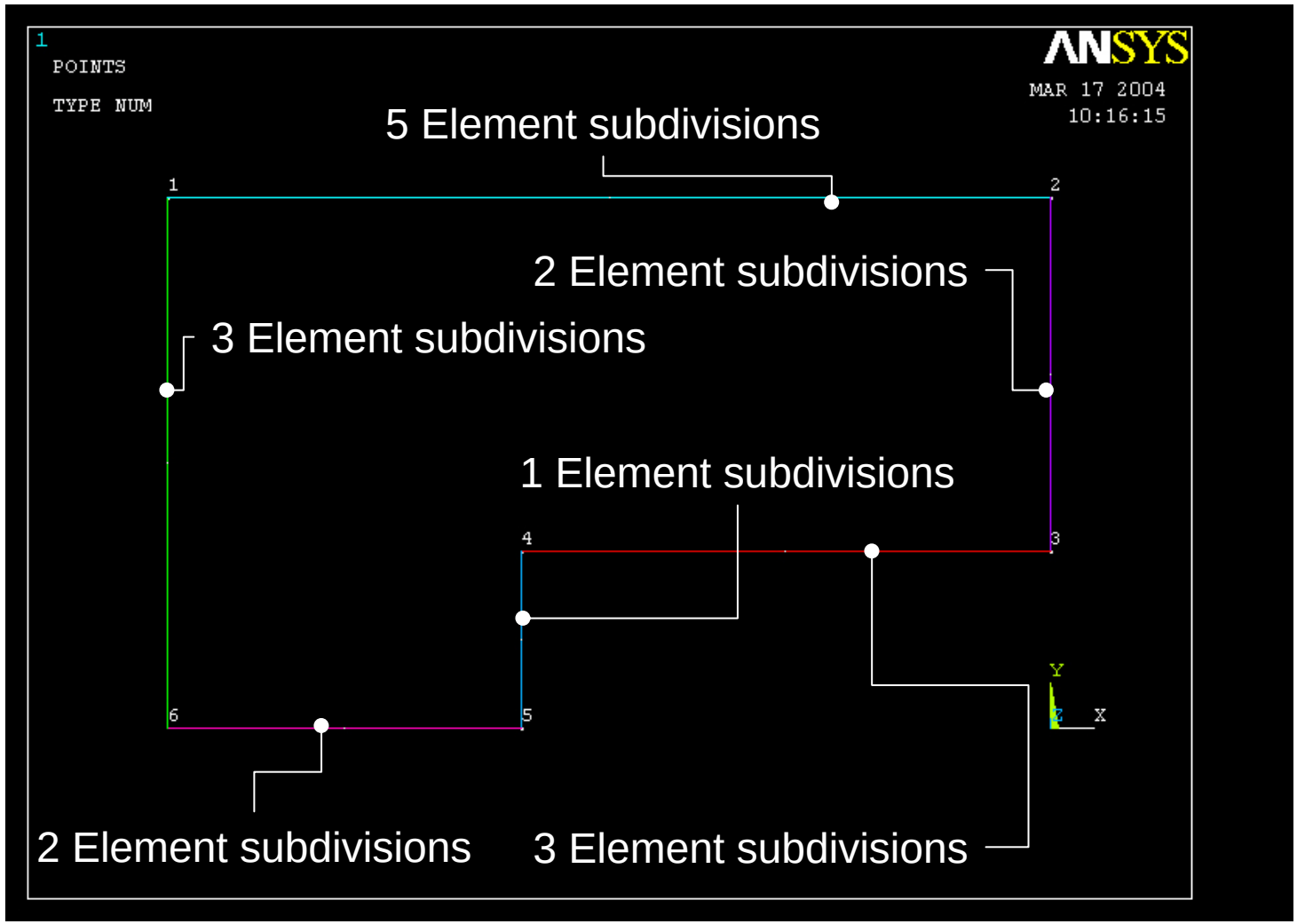
Select/Pick Lines to specify mesh size for

Pick the two longest lines

See next page

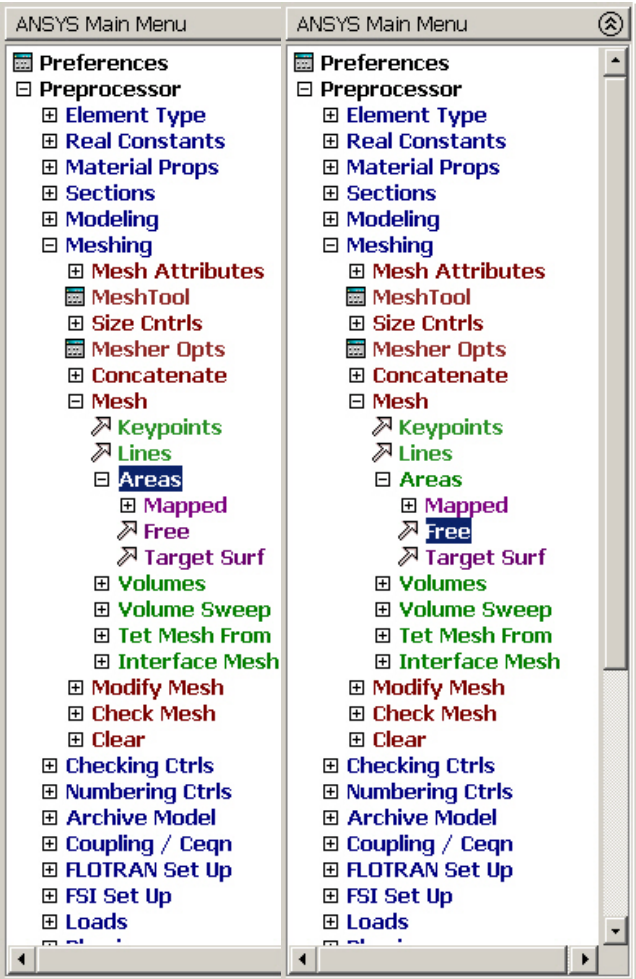
Press OK when finish with selection

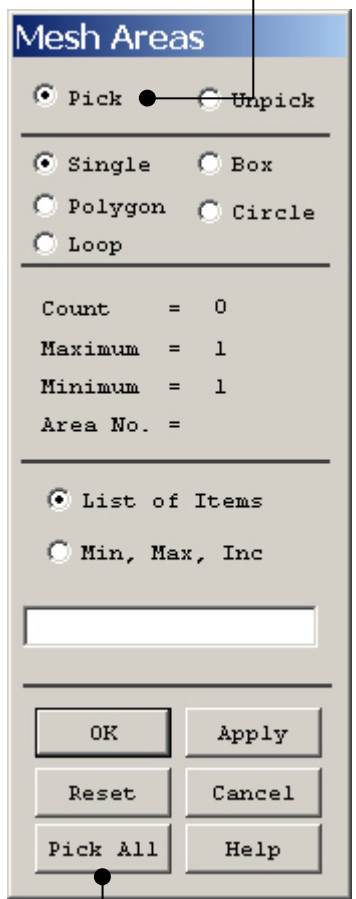
Example - Lines



Example – Free Meshing

Preprocessor > Meshing > Mesh > Areas > Free





Select individual areas to be meshed

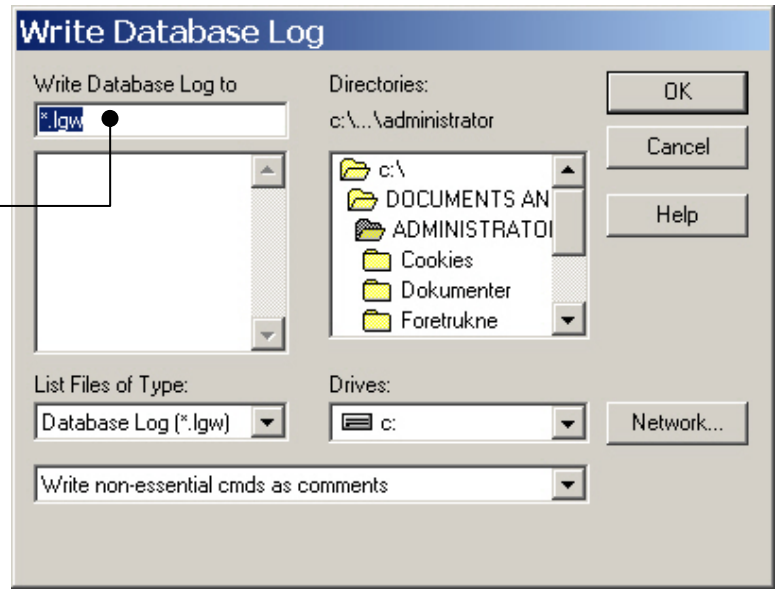
NB: It is often necessary to “Clear” the model for example if Element Type or model geometry is to be changed

Select all areas defined to be meshed

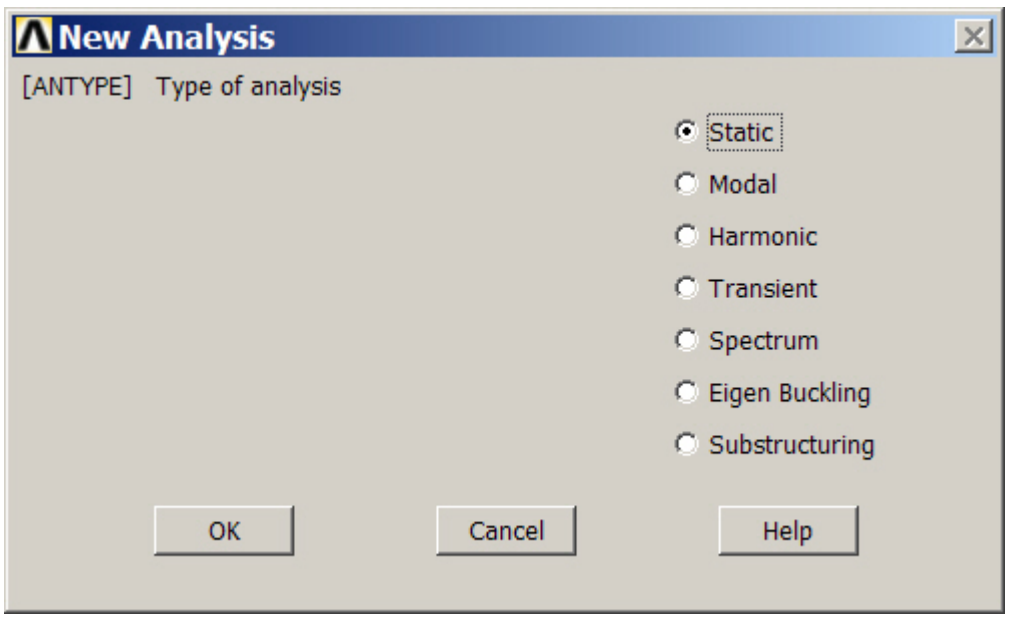
Example – Analysis Type

File > Write DB log file

Enter “example0601.lgw”

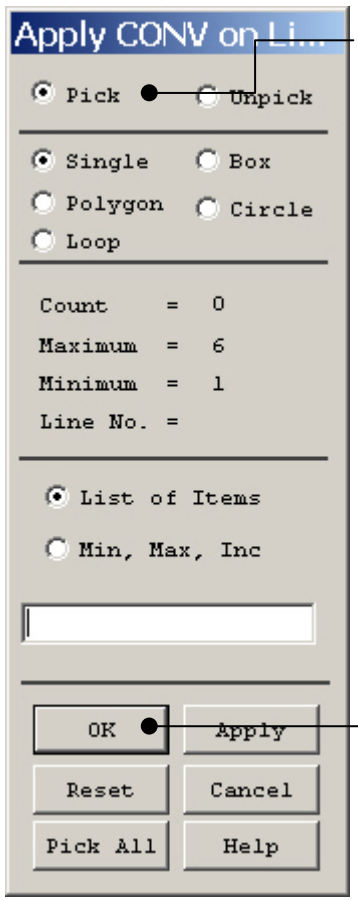


Solution > Analysis Type > New Analysis



Example – Define Loads

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

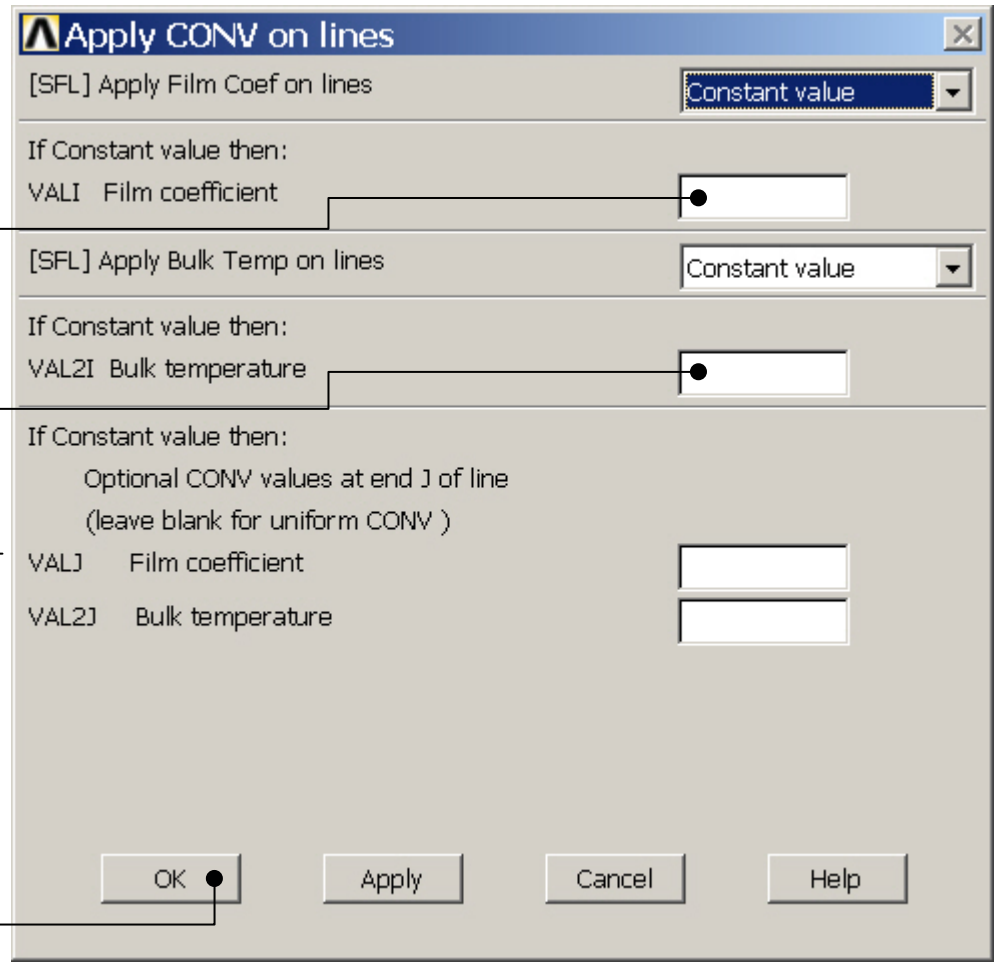


Select the top horizontal line L1

Enter 1000

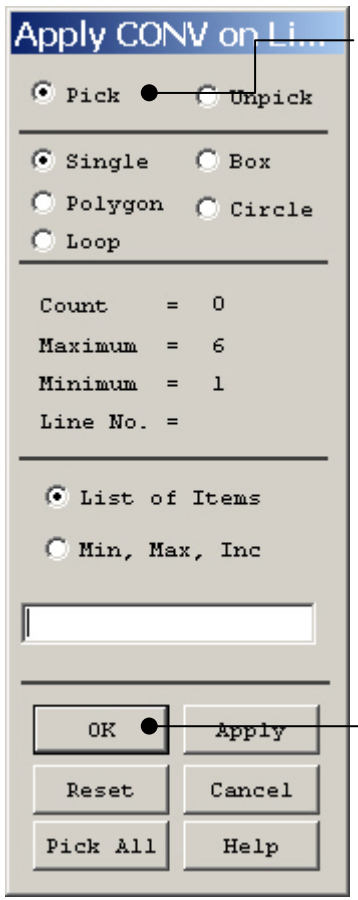
Enter 1700

Press OK



Example – Define Loads

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

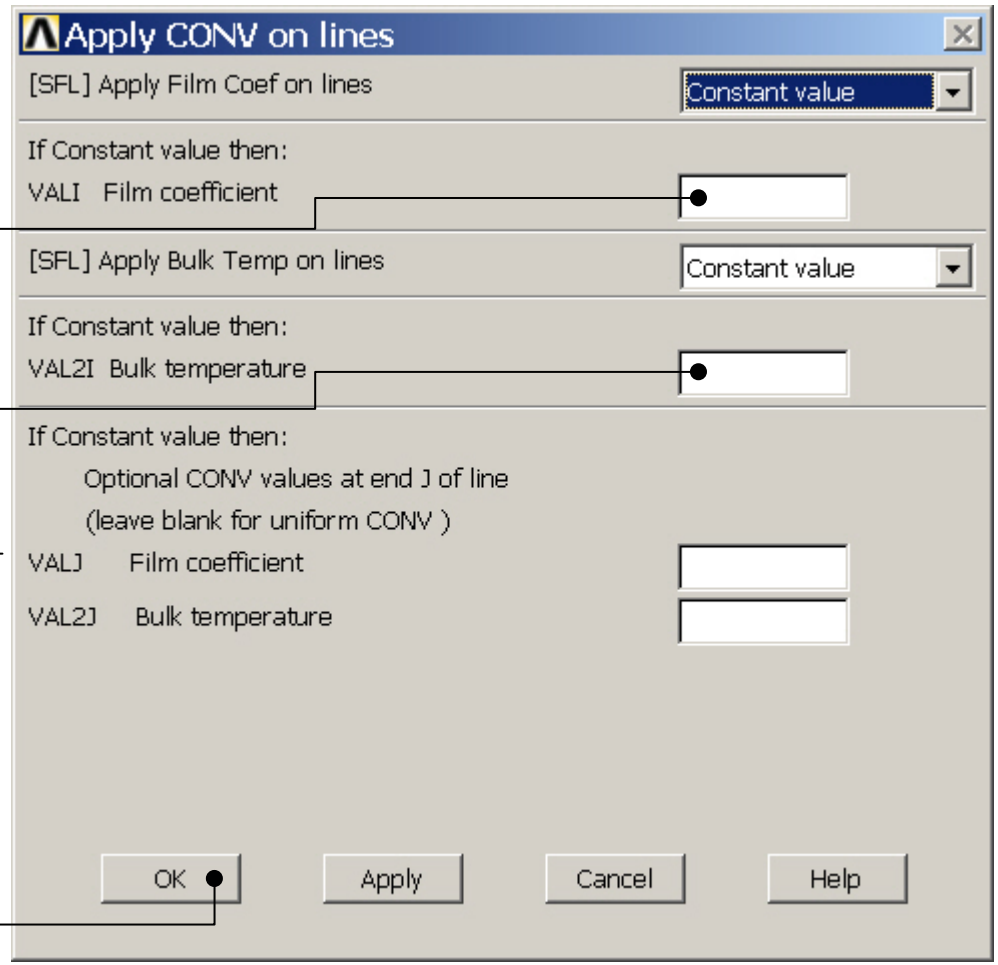


Select the lines L3 and L4

Enter 200

Enter 400

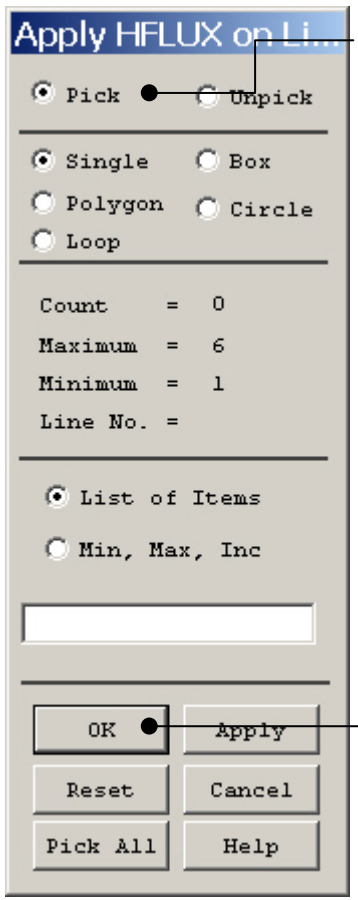
Press OK

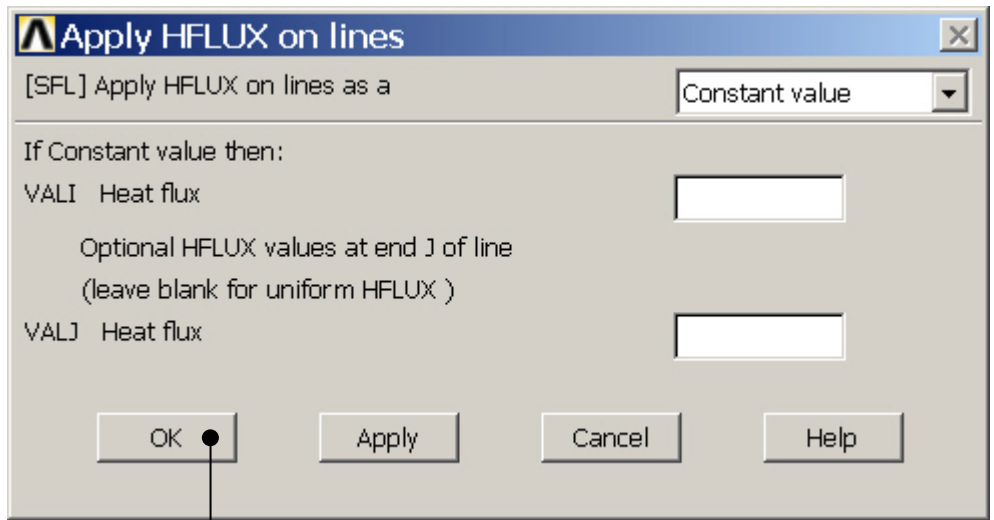


Example – Define Loads

Solution > Define Loads > Apply > Thermal > Heat Flux > On Lines

Select the lines L6, L2 and L5

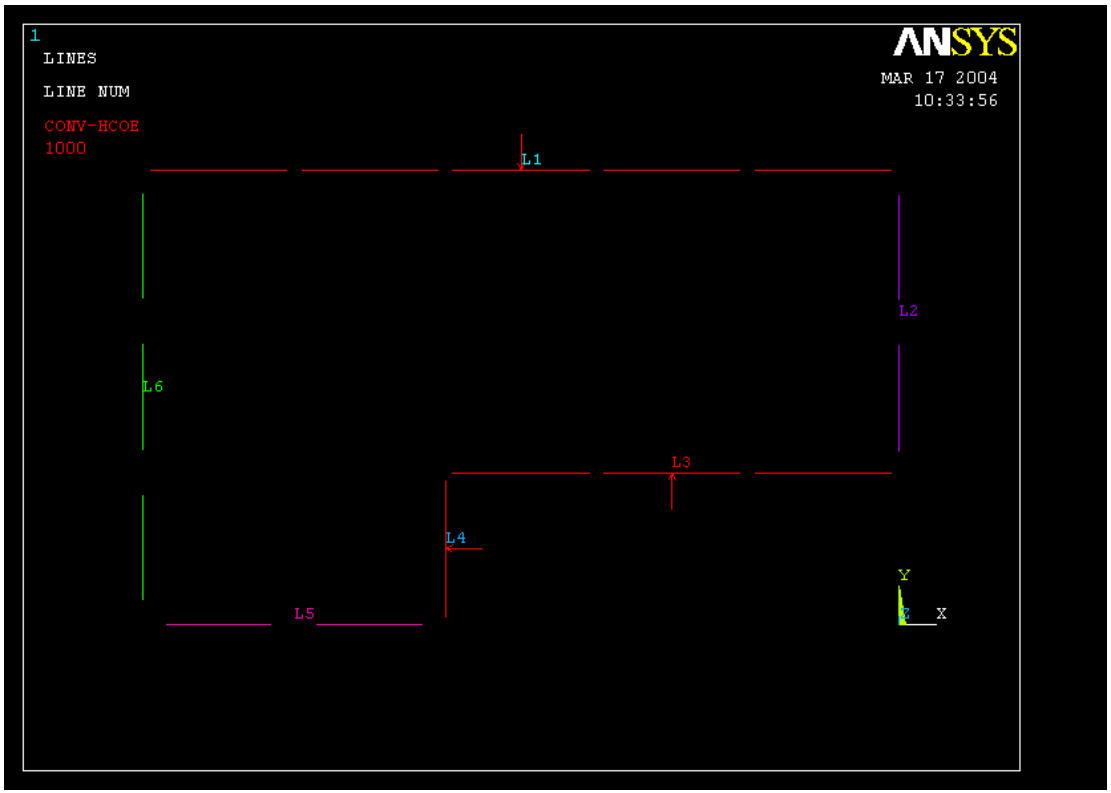




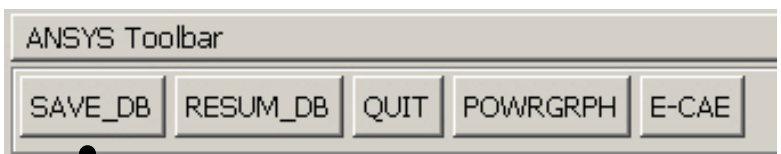
Press OK

By default, this will specify a heat flux of zero along these three lines, which is consistent with the symmetry of the problem.

Example - Save



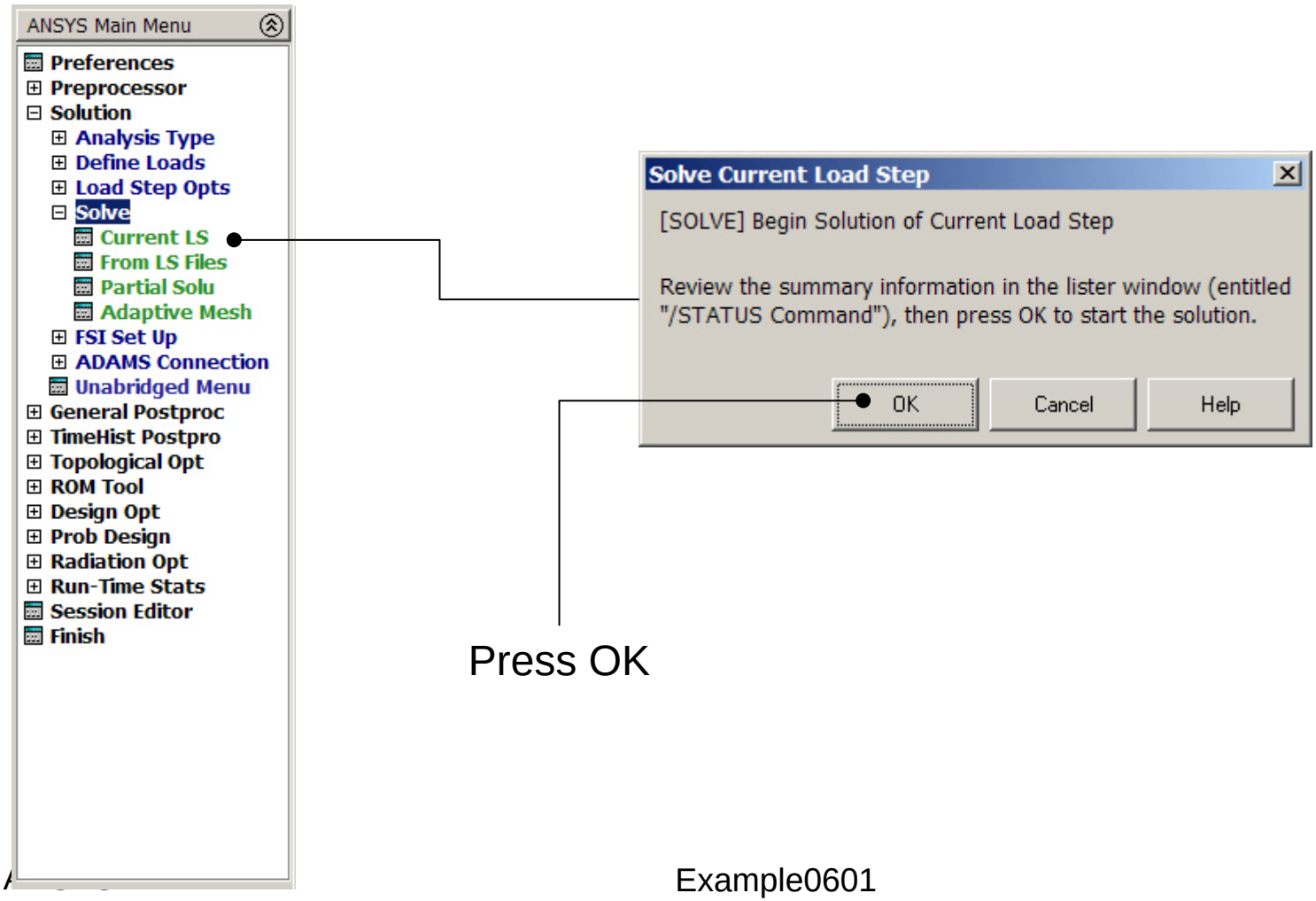
Display of Analysis model



Save the model

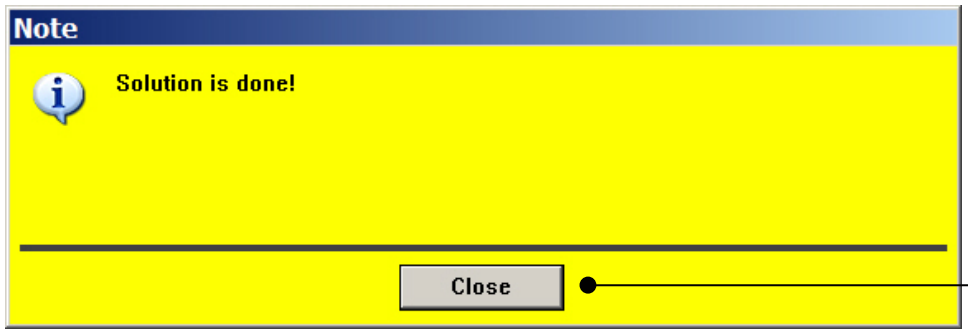
Example - Solve

Solution > Solve > Current LS



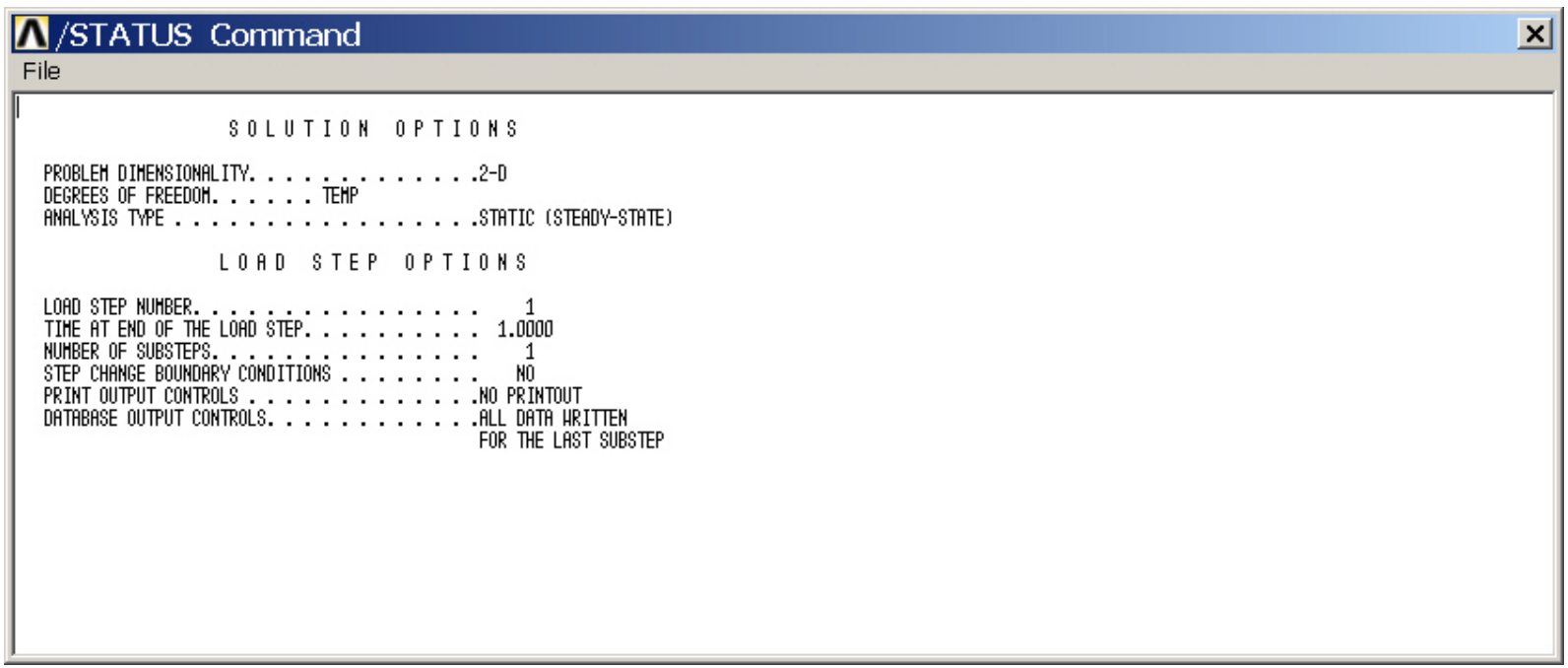
Press OK

Example - Solve



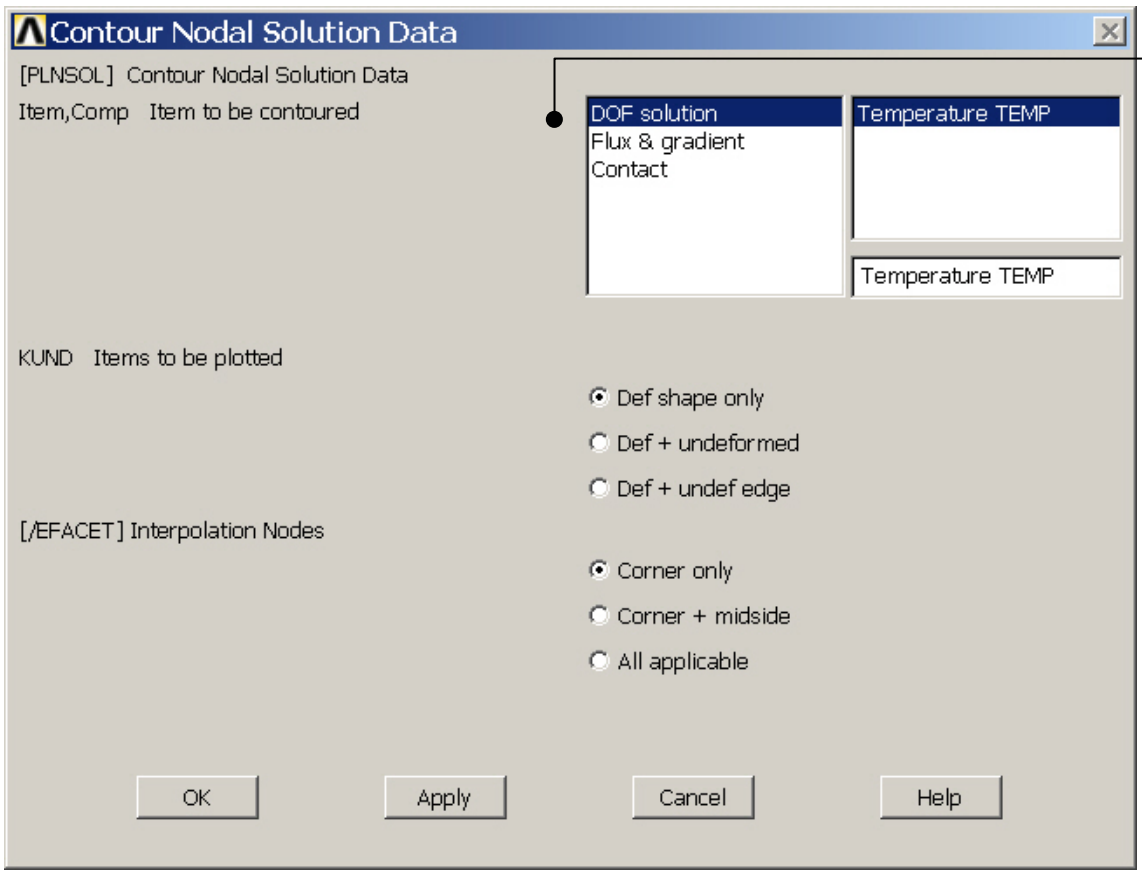
Press Close

Press here
to Close



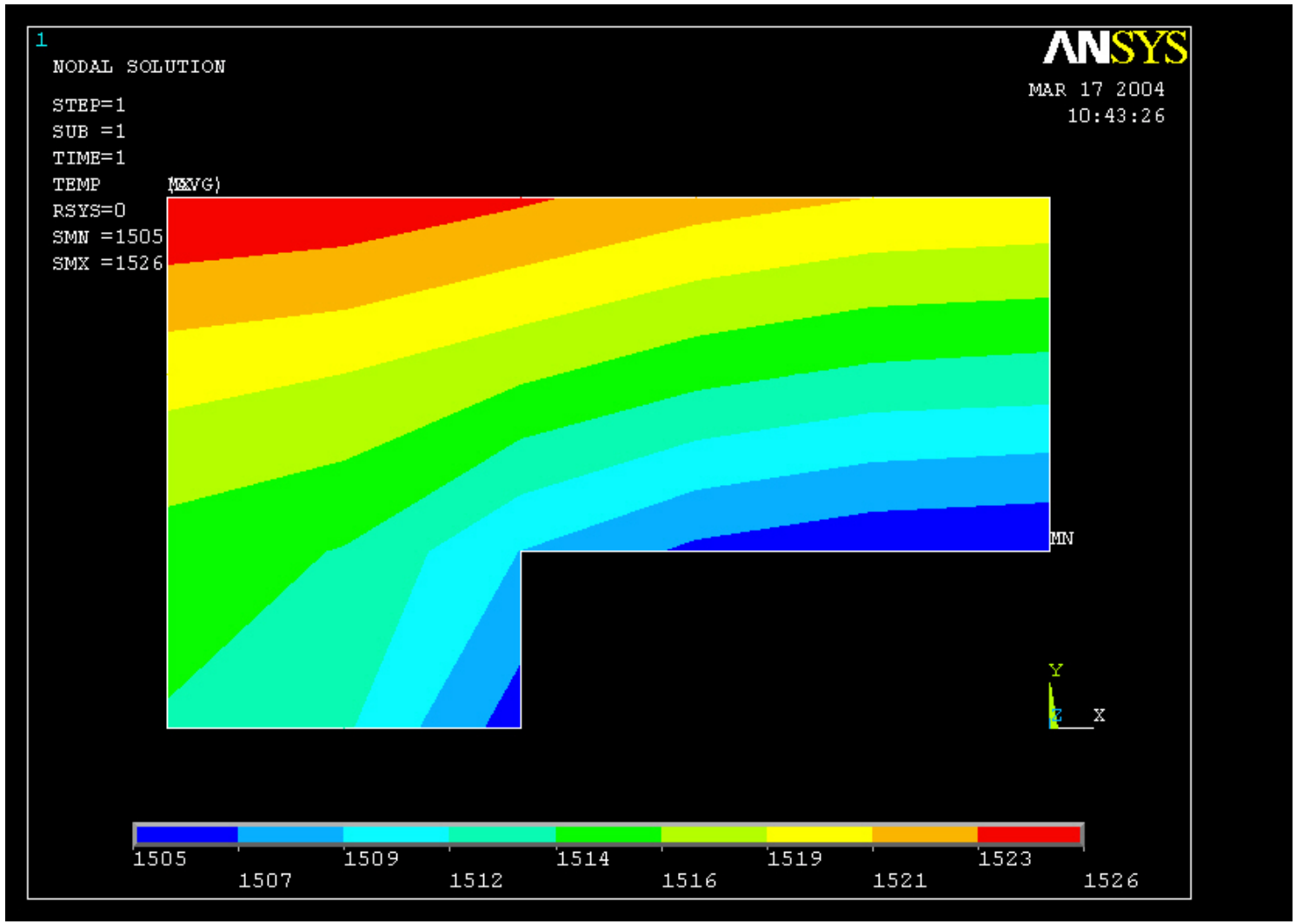
Example – Contour Plot

General Postproc > Plot Results > Contour Plot > Nodal Sol

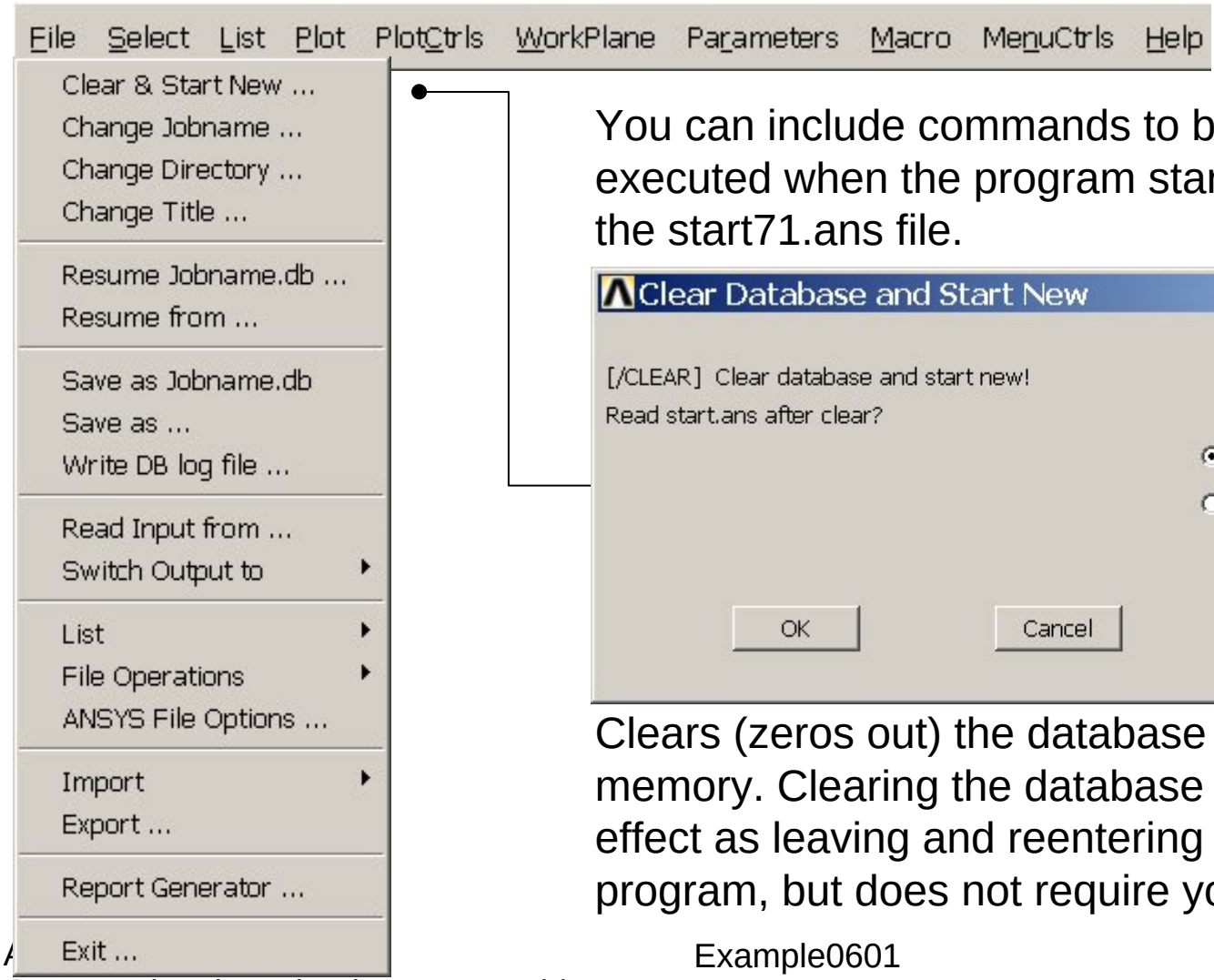


Select DOF solution

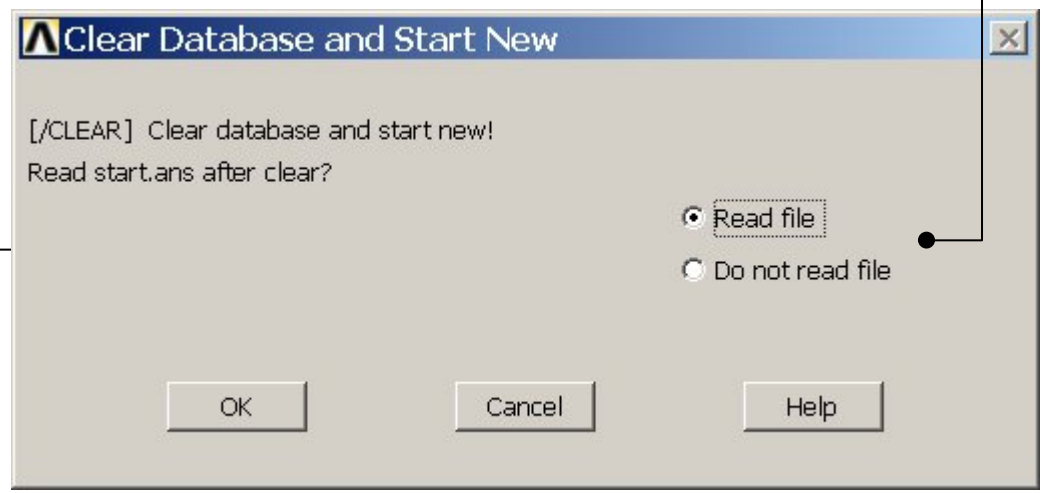
Example – Contour Plot



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.