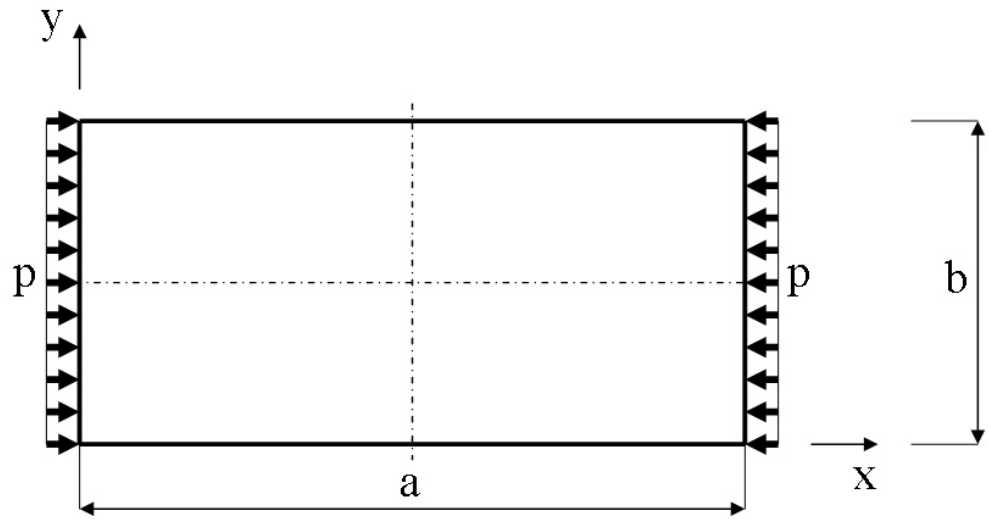
Course in ANSYS

Example0541

Example – Plate



Objective:
Plot the P-U curve for the nonlinear behaviour
Tasks:
Model the geometry
Run a static linear analysis with Prestress On
Run an eigen-buckling analysis
Run the nonlinear analysis

$E = 210000 N/mm^2$
$\nu = 0.3$
a = 200mm
b = 100mm
t = 1mm
p = 70N/mm

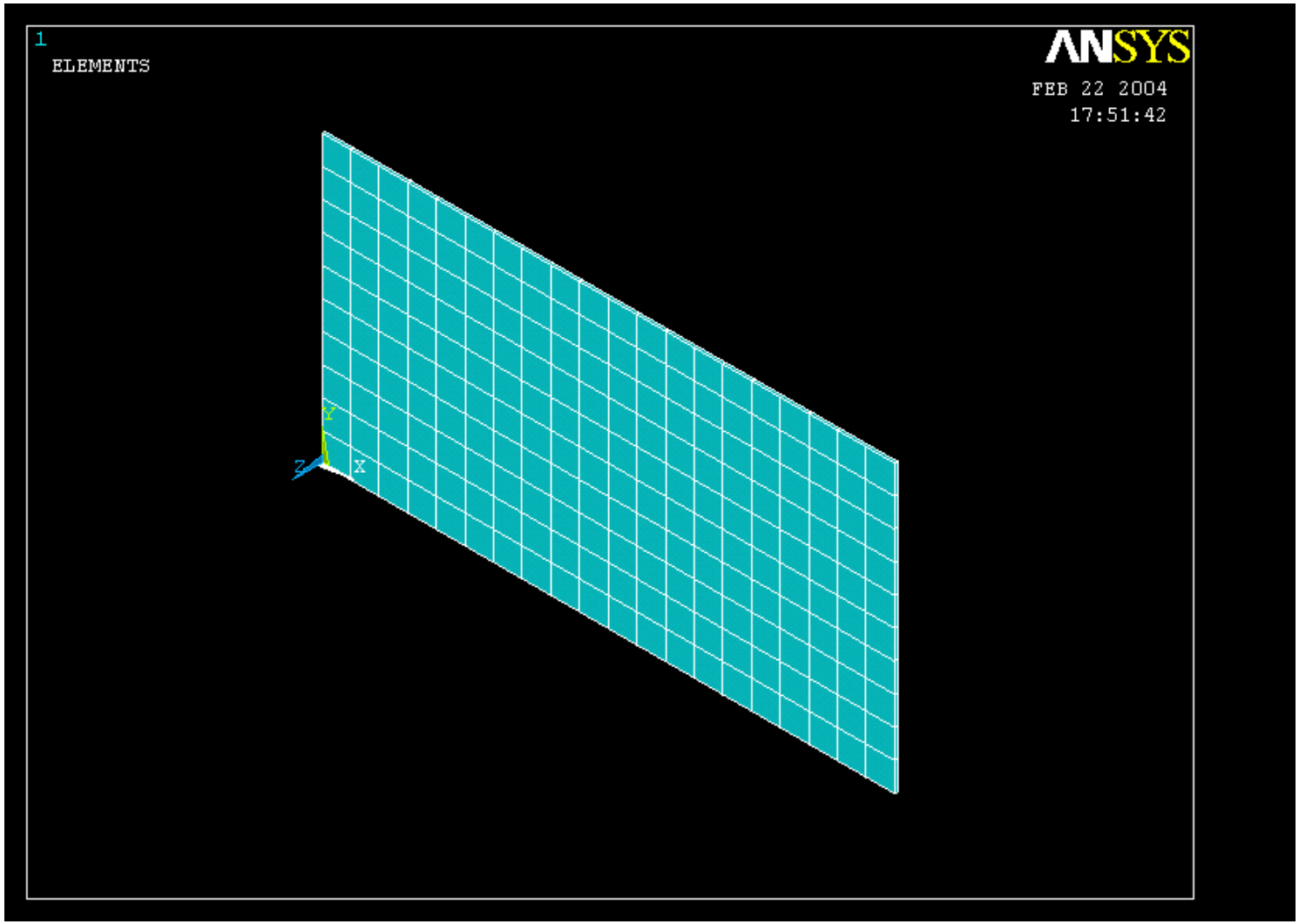
Example – Plate

- Nonlinear buckling analysis in ANSYS is somewhat simpler than eigenvalue buckling analysis since there is only one solution step; however, it can require more than one load step in solution. In general, a nonlinear buckling analysis is simply a nonlinear static analysis in which the load is increased until the solution fails to converge, indicating that the structure cannot support the applied load (or that numerical difficulties prevent solution). If the structure does not lose its ability to support additional load when it buckles (the plate we are analyzing is an example of such a structure), a nonlinear buckling analysis can also be used to track post-buckling behavior. We will not carry out post-buckling analysis here because of uncertainty in the validity of the results.

Example – Read input from

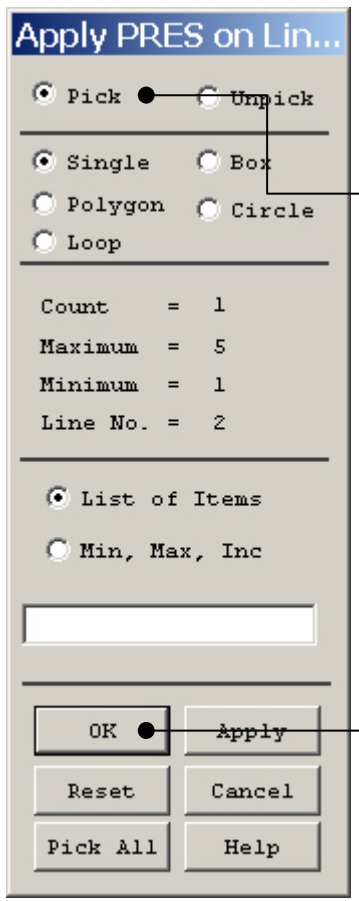
Load the example0505.lgw by **File Menu > Read input from**

Example - Plate



Example – Define Loads

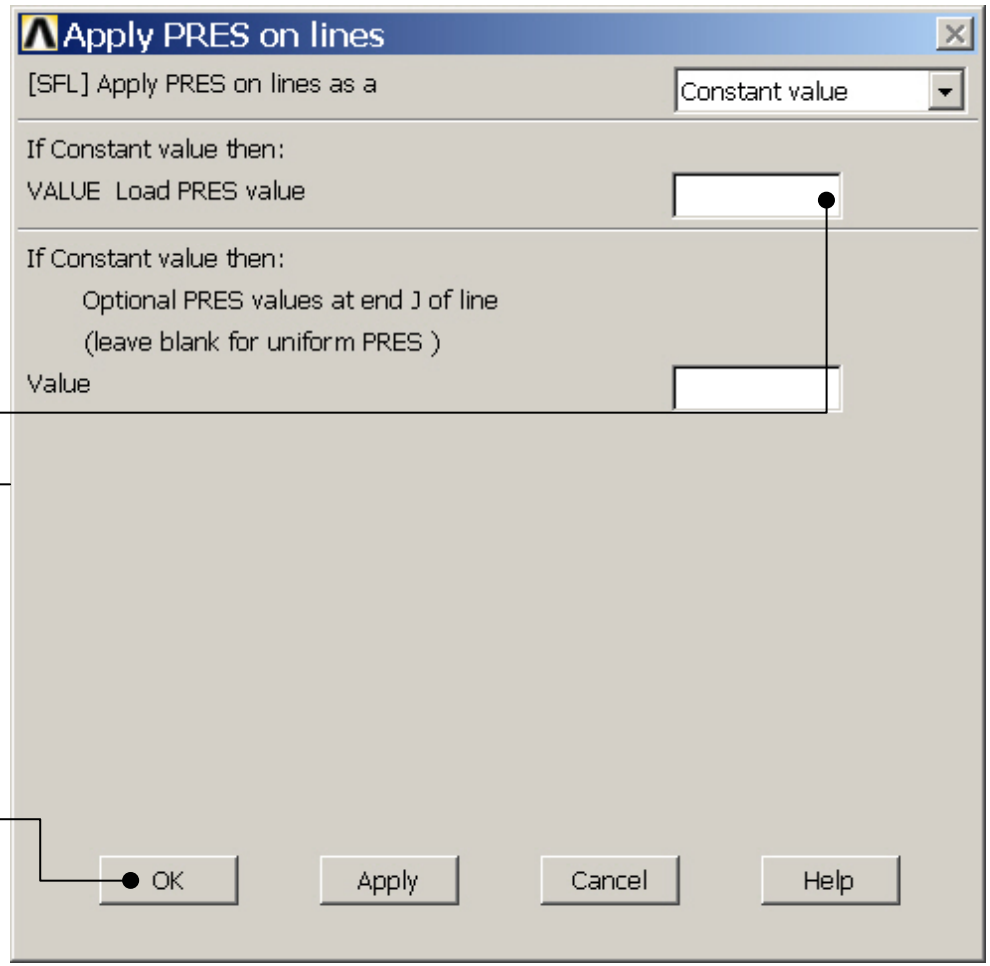
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the left and right straight line

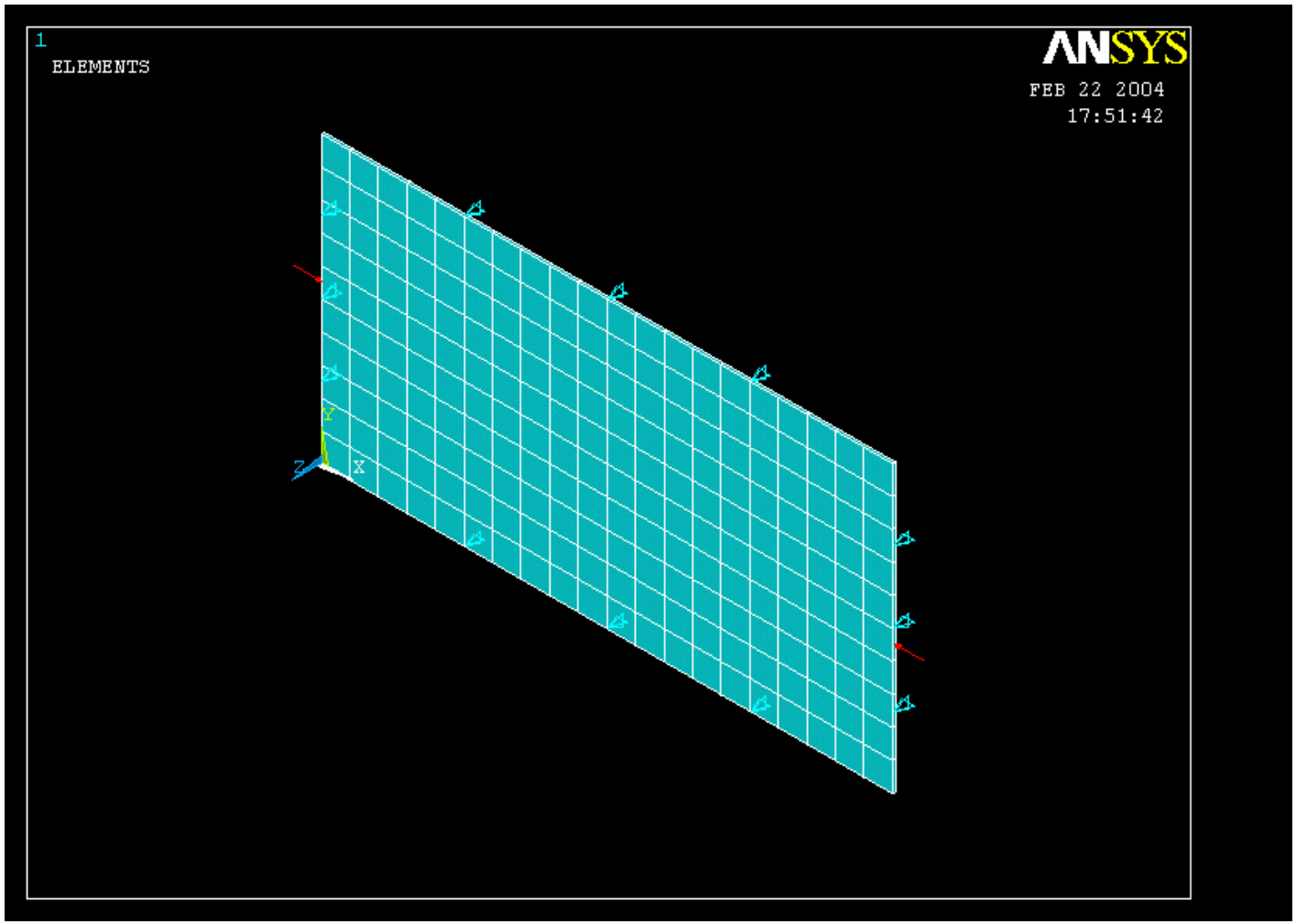
Enter 70

Press OK to finish



Note: Pressure acts normal and inward to a surface

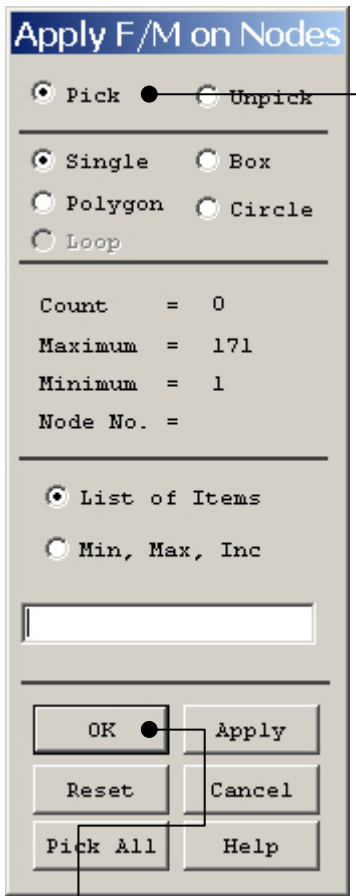
Example - Plate

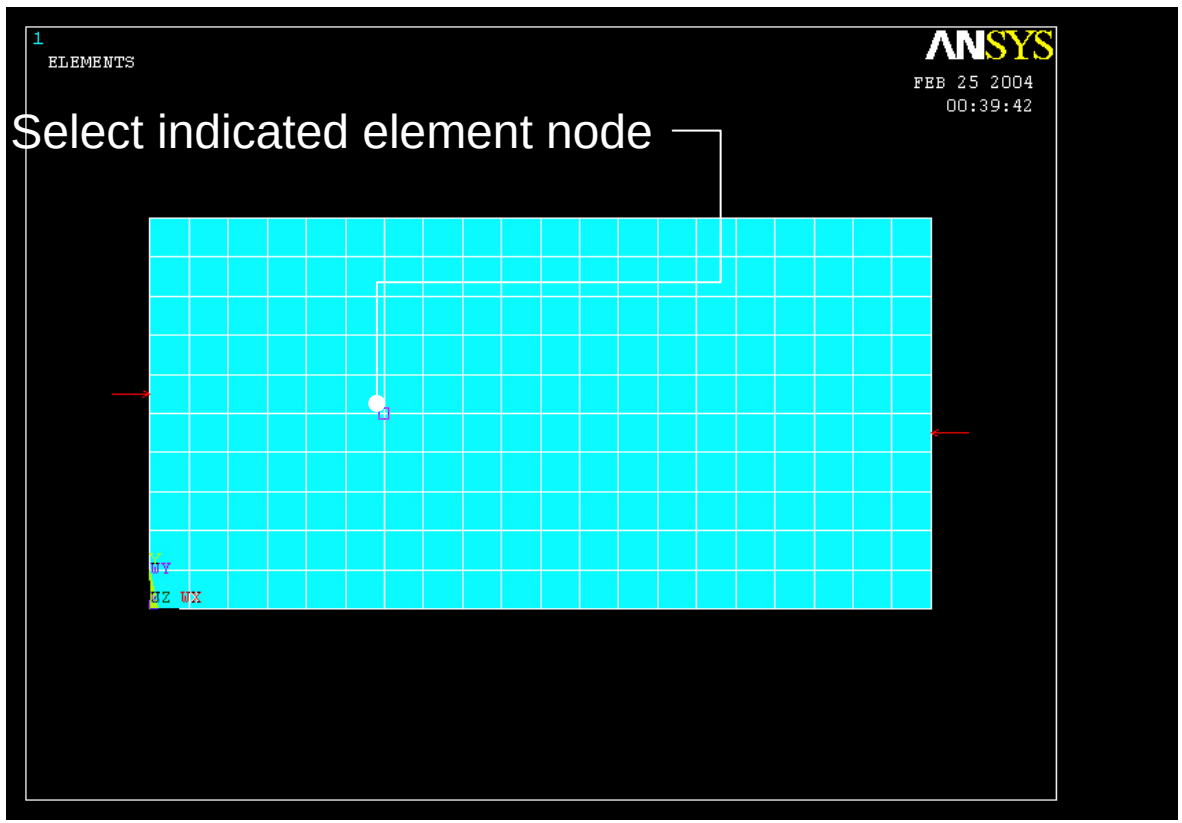


Example – Define Loads

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

Note: If the model is remeshed all loads will be deleted with the element nodes



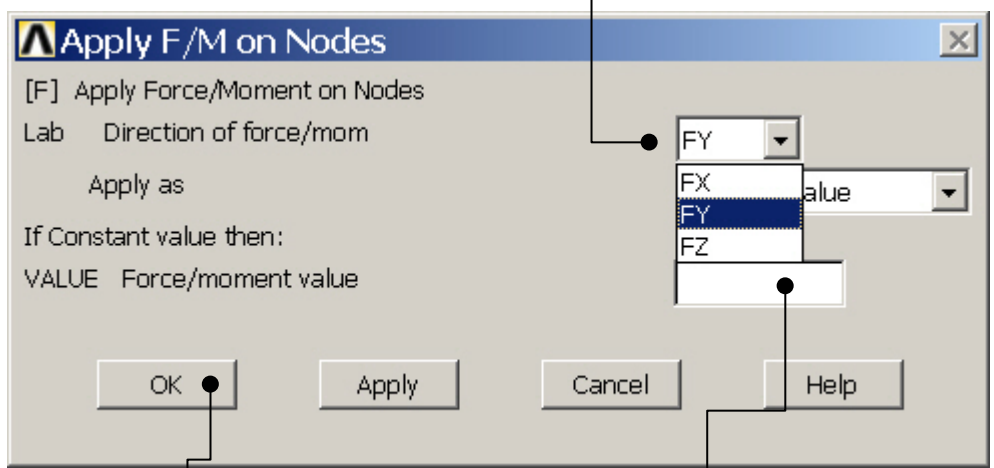


Press OK

Example – Define Loads

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

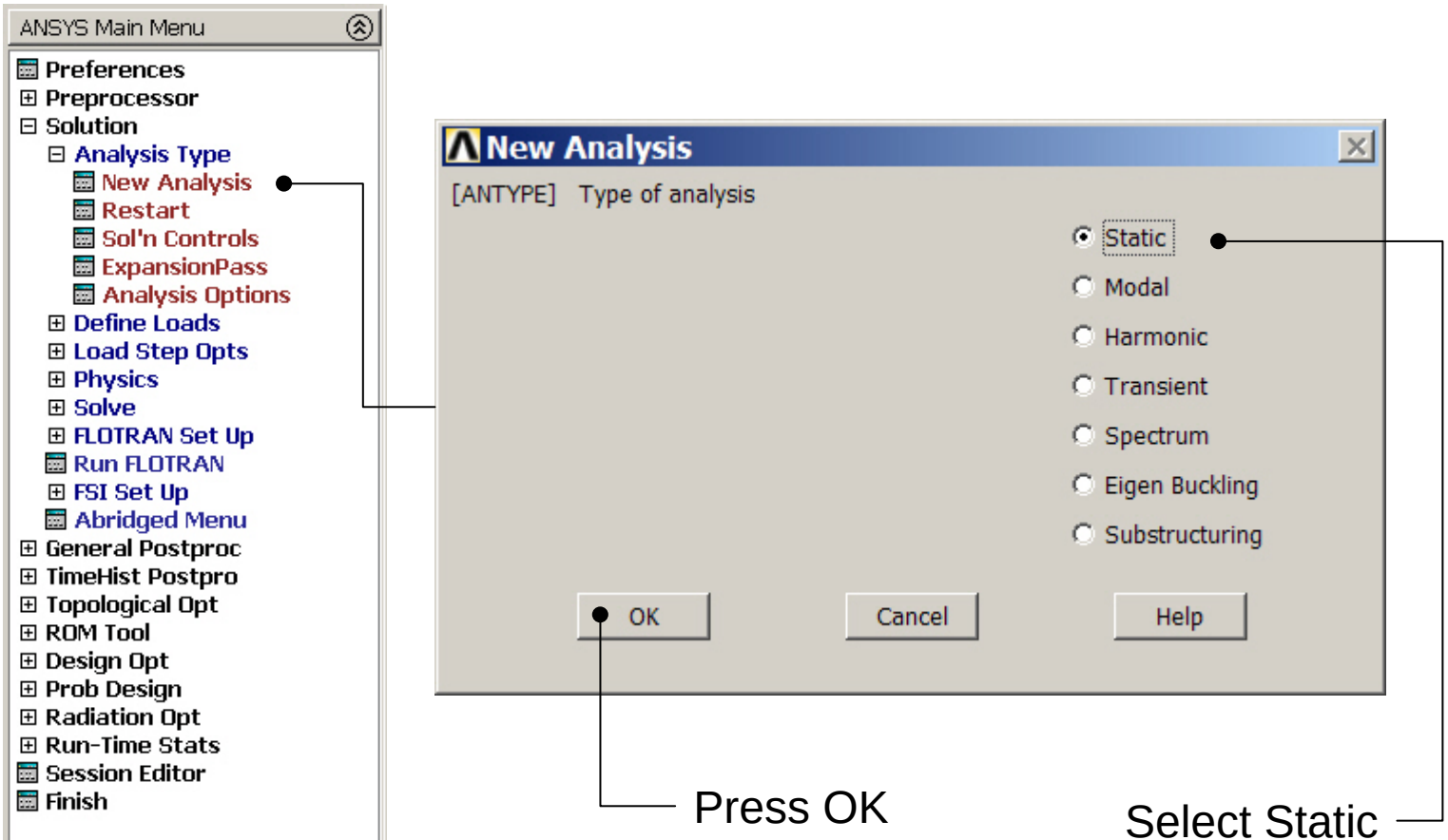
Change to FZ



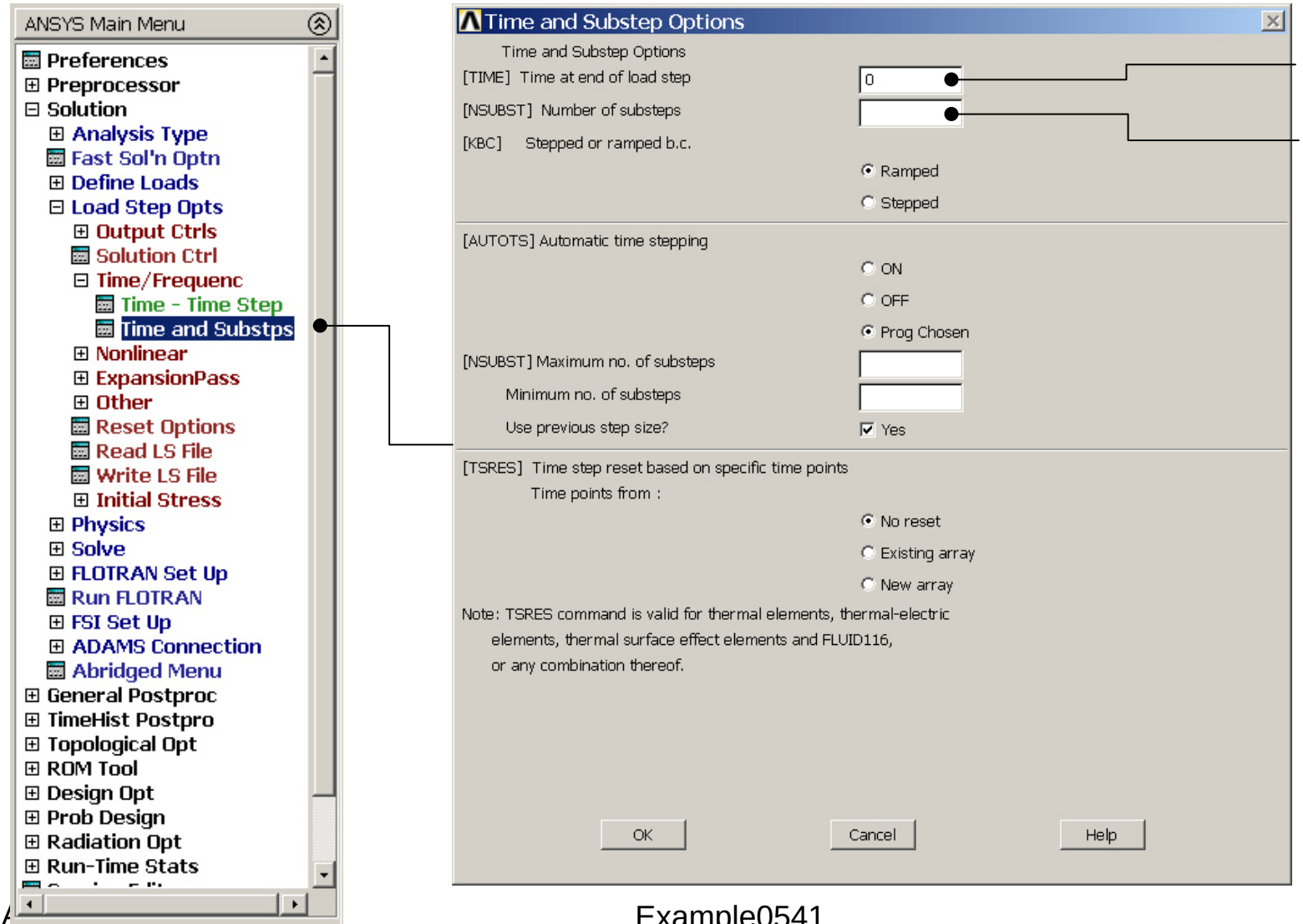
Press OK to finish

Enter 0.1

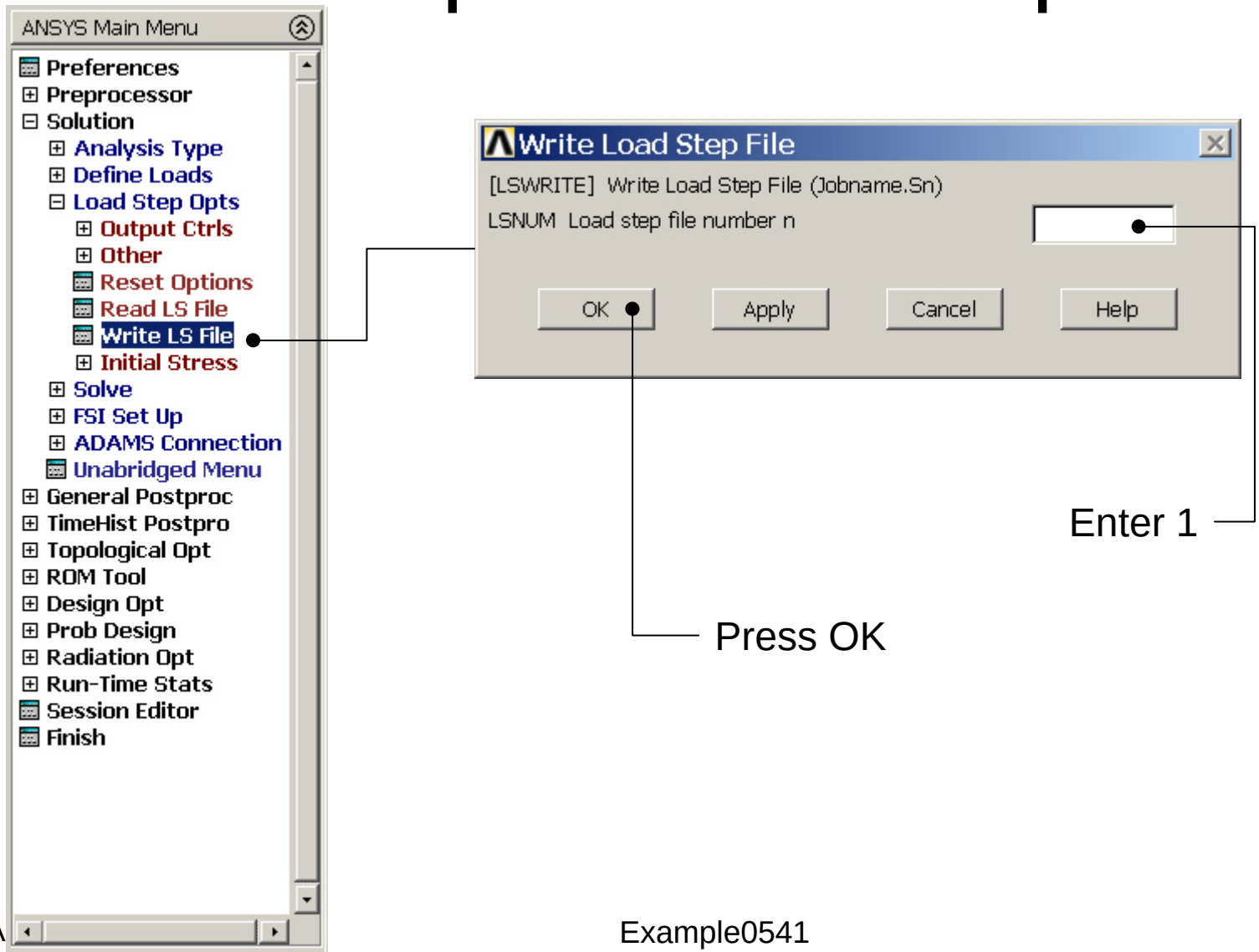
Example – Analysis Type



Example – Load Step Opts

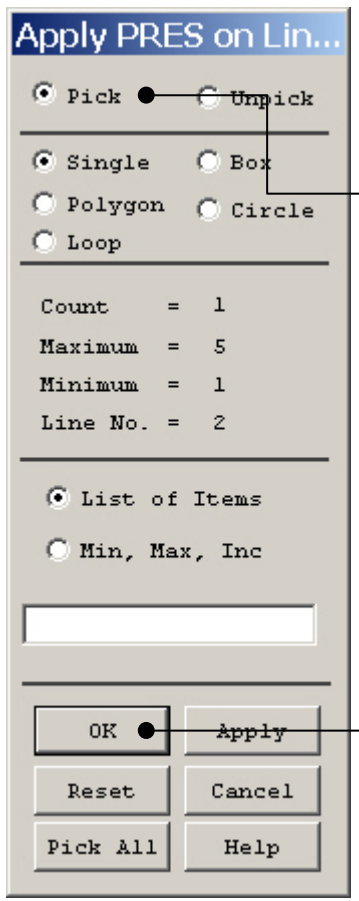


Example – Loadstep file



Example – Define Loads

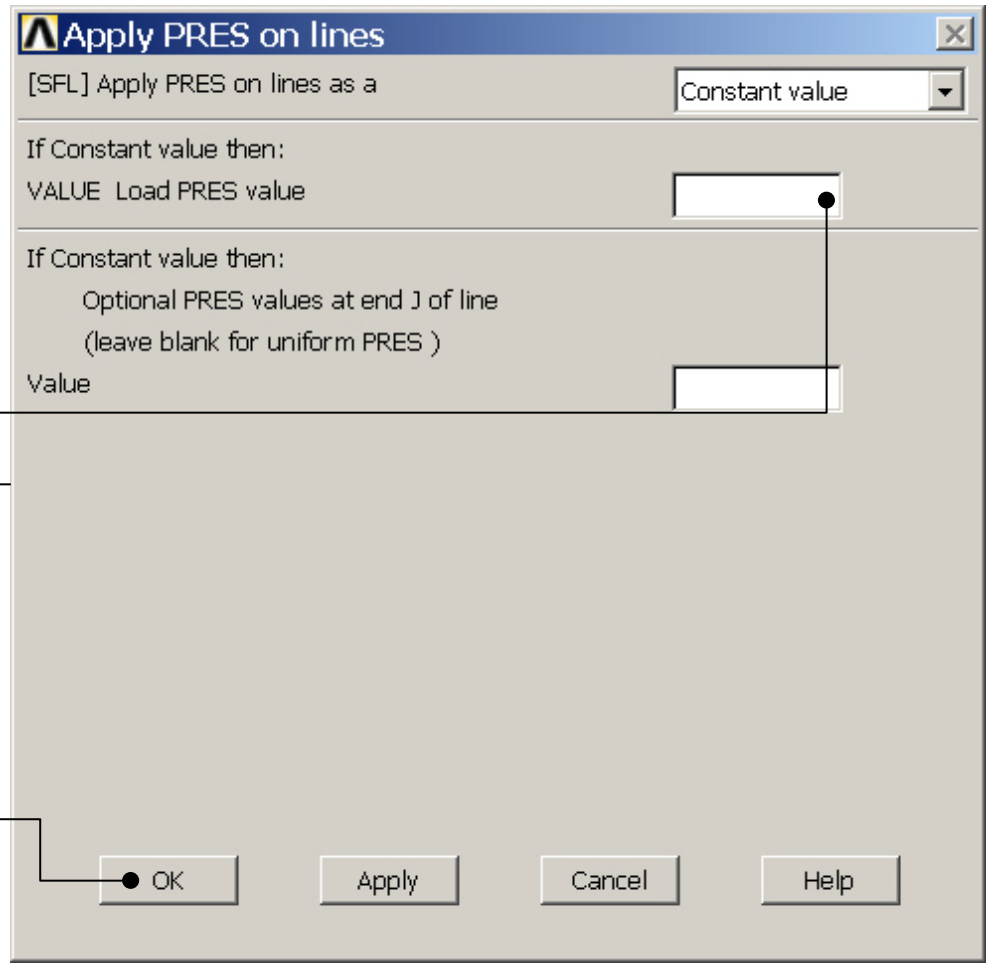
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the left and right straight line

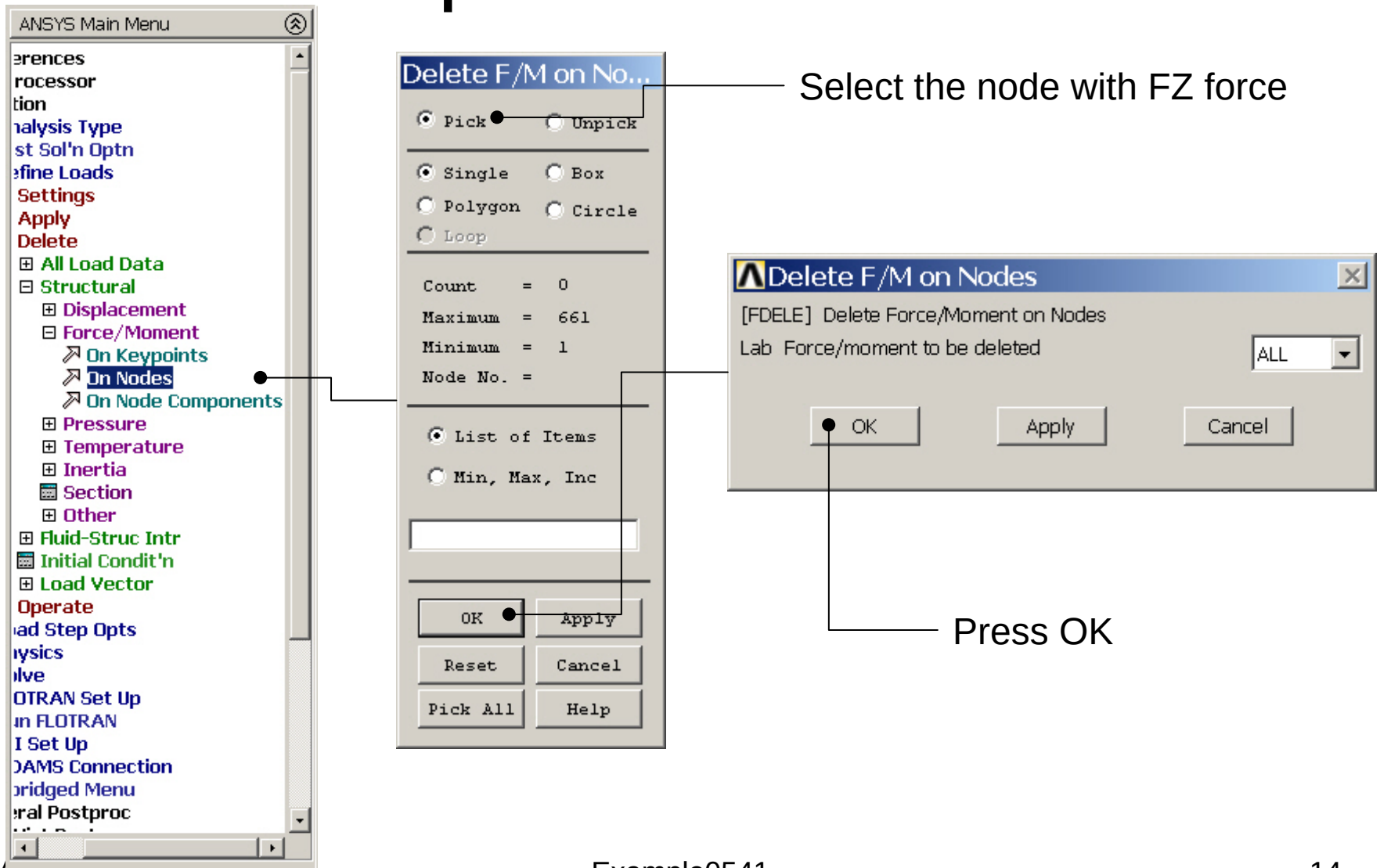
Enter 80

Press OK to finish

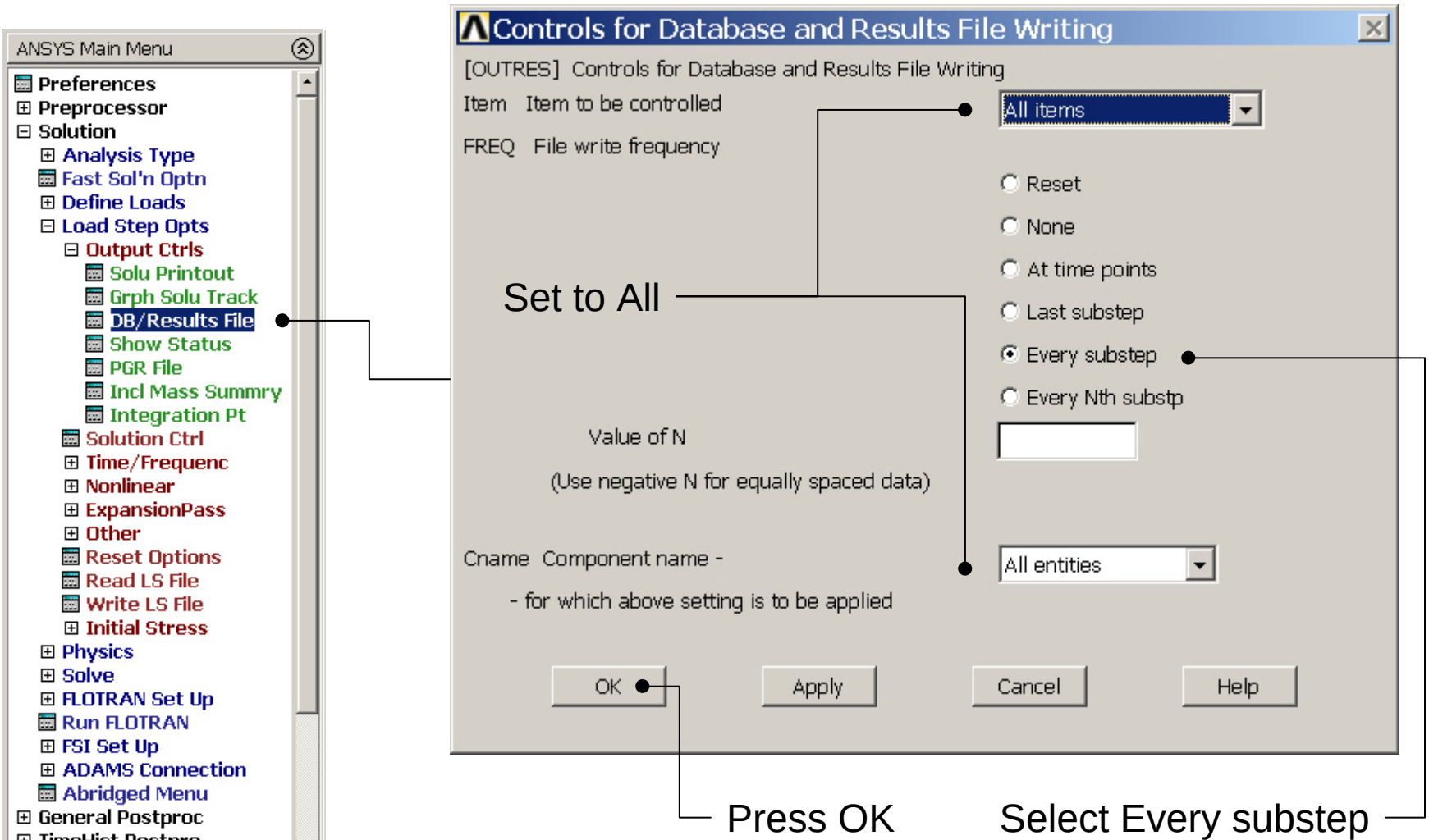


Note: Pressure acts normal and inward to a surface

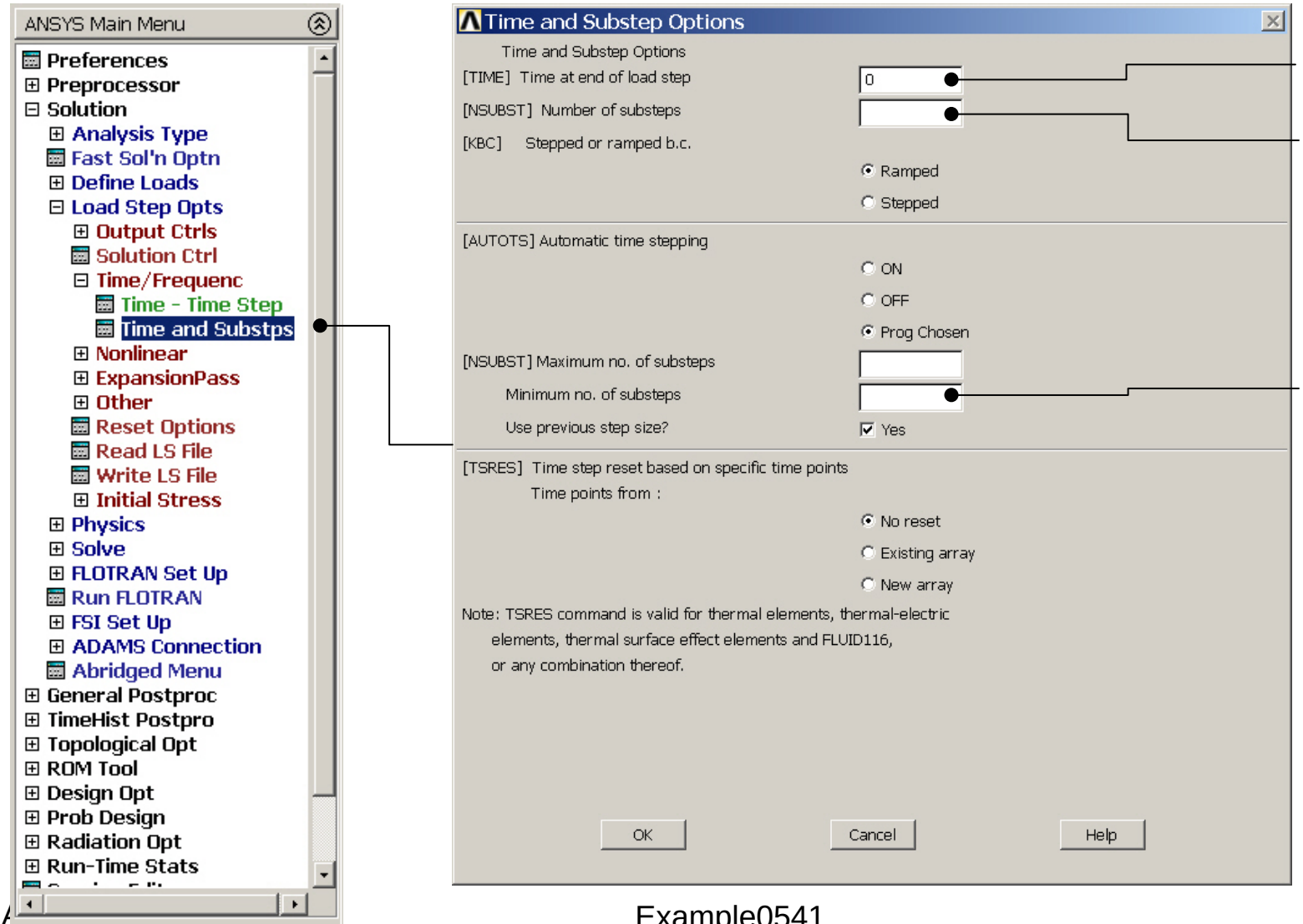
Example – Delete Load



Example – Output Ctrls

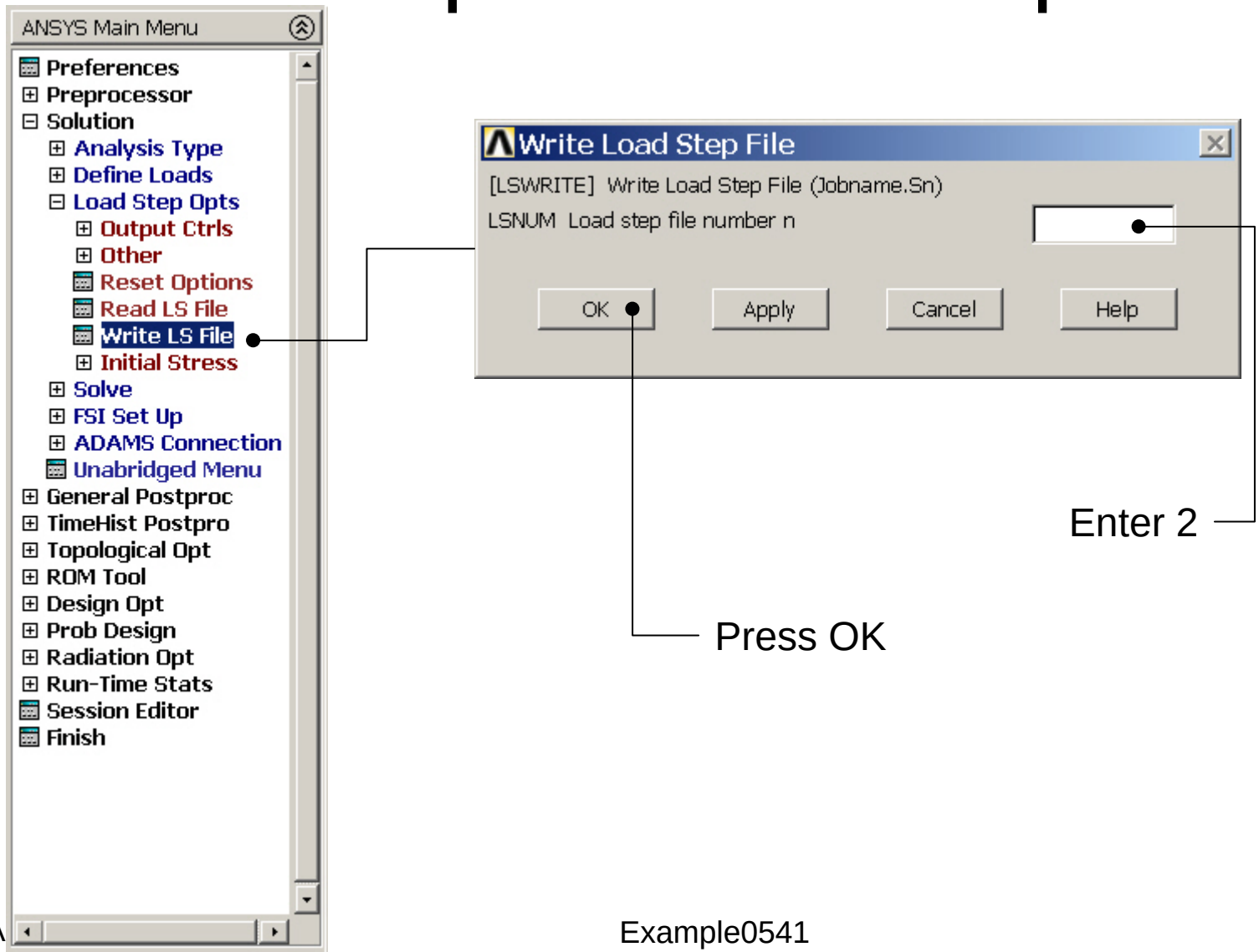


Example – Load Step Opts



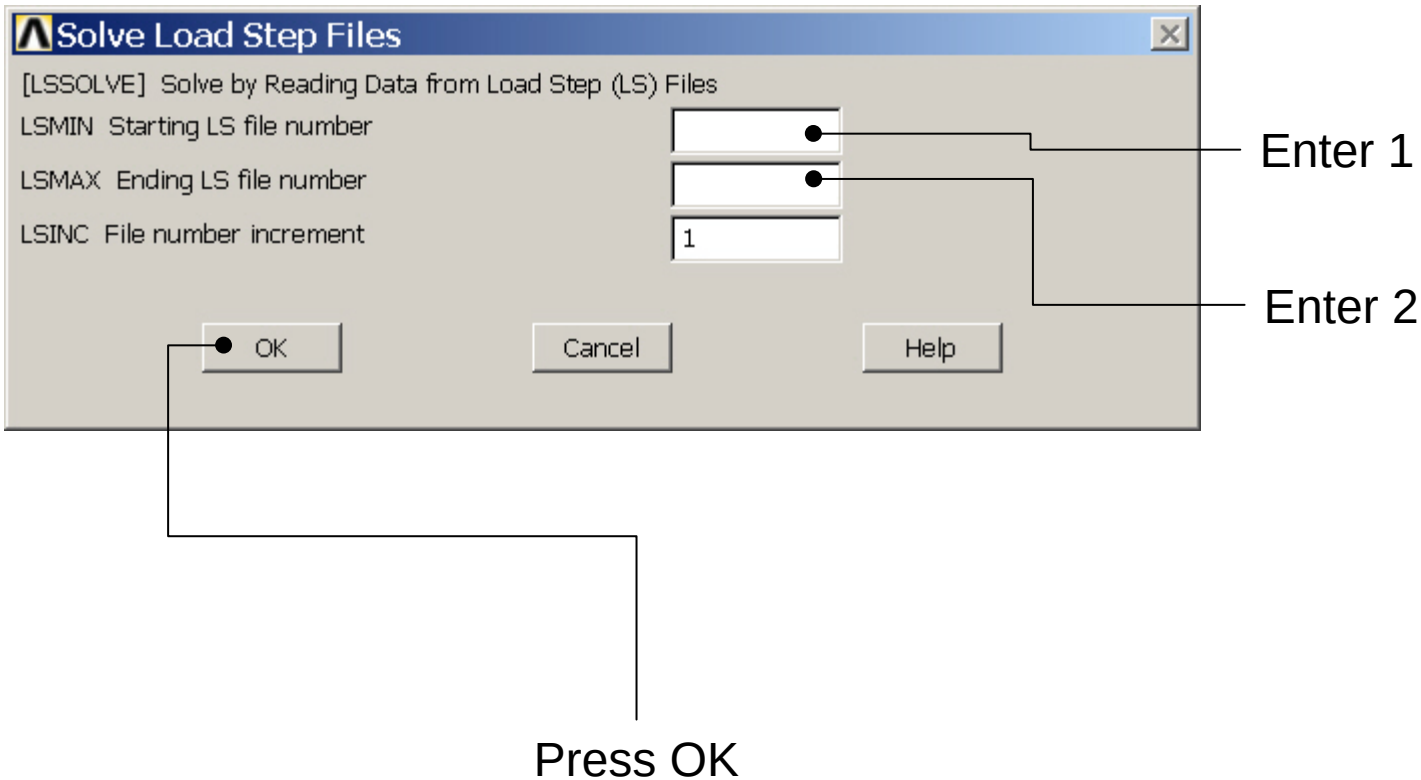
In case ANSYS decides to perform automatic time stepping

Example – Loadstep file

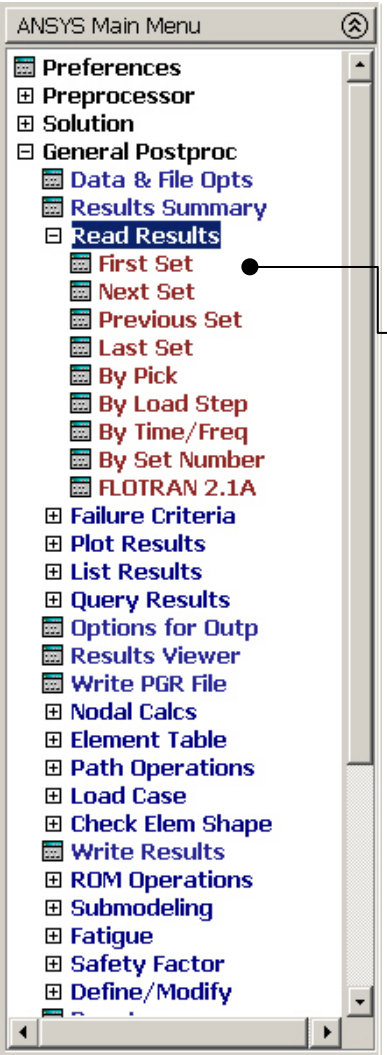


Example - Solve

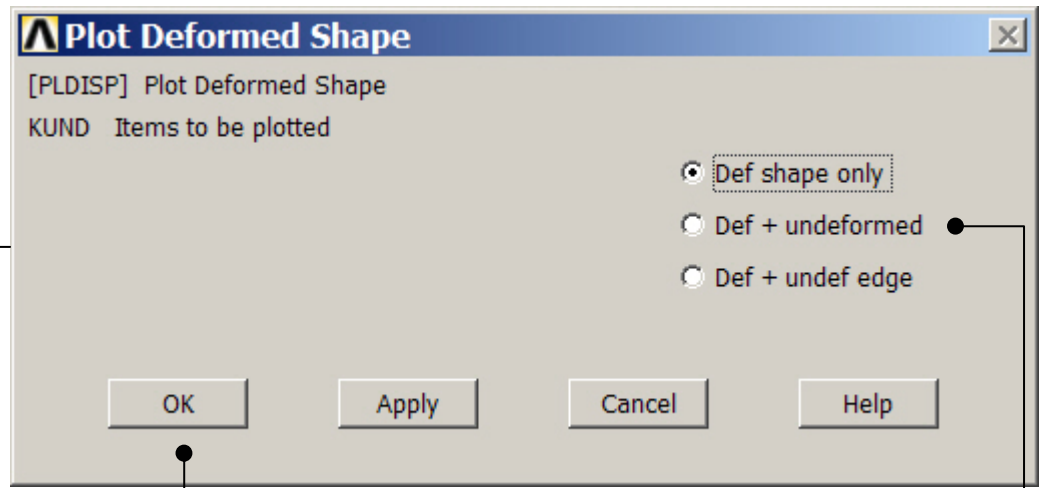
Solution > Solve > From LS Files



Example – Read Results

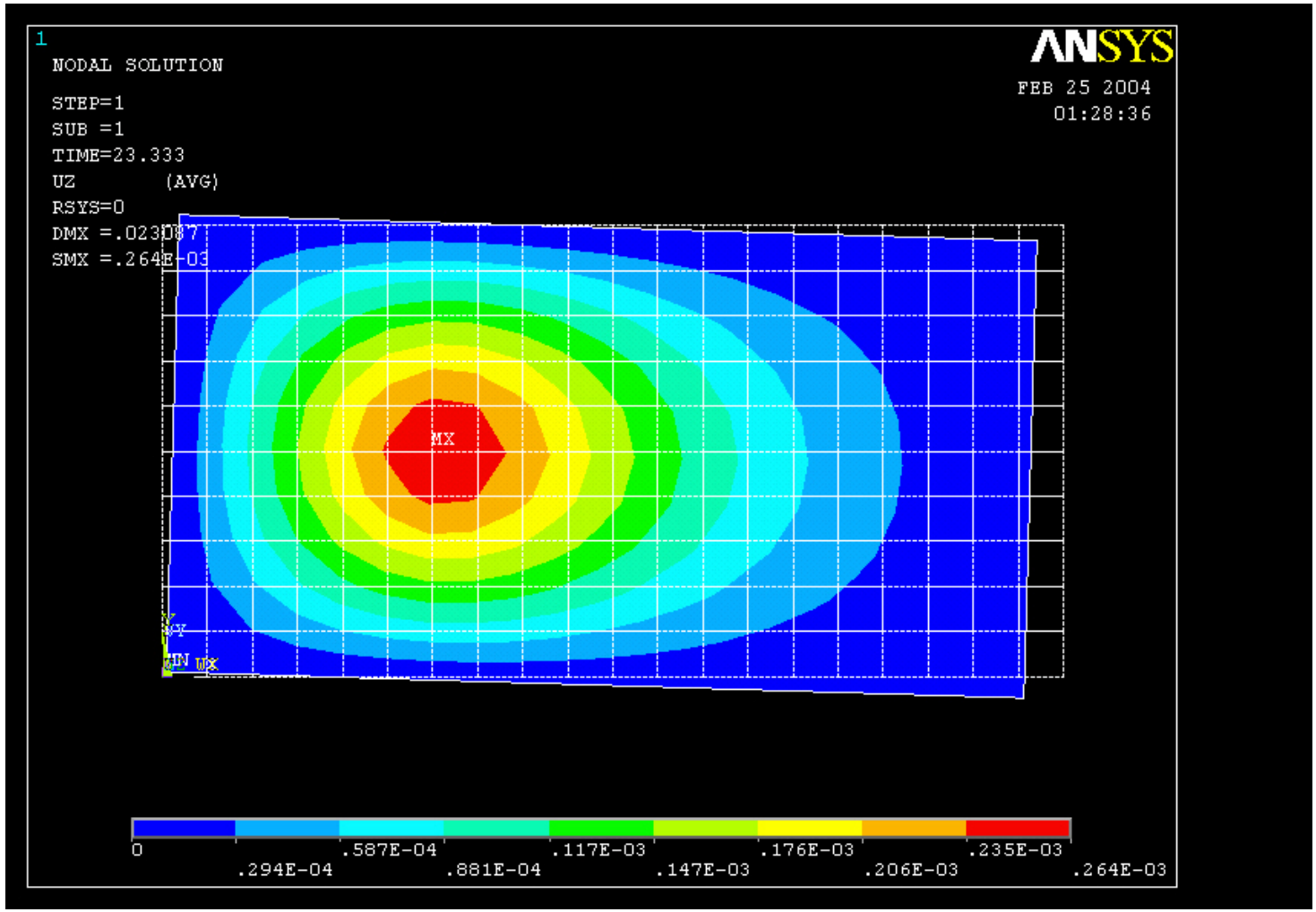


General Postproc > Plot Results > Deformed Shape

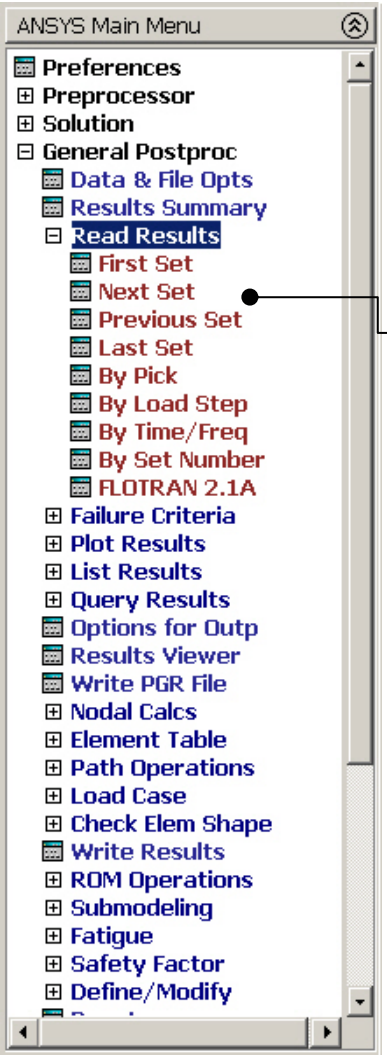


Select “Def+undeformed” and Press OK

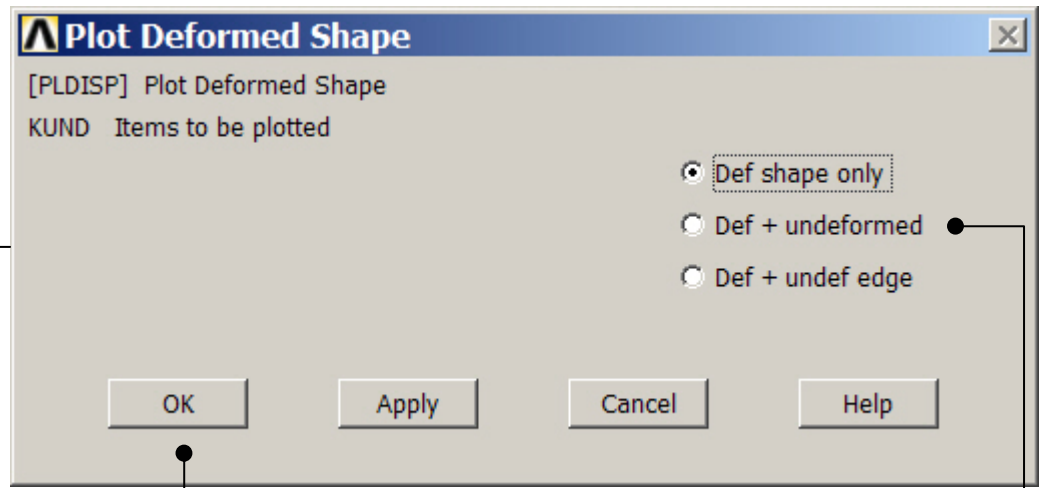
Example – Contour Plot



Example – Read Results

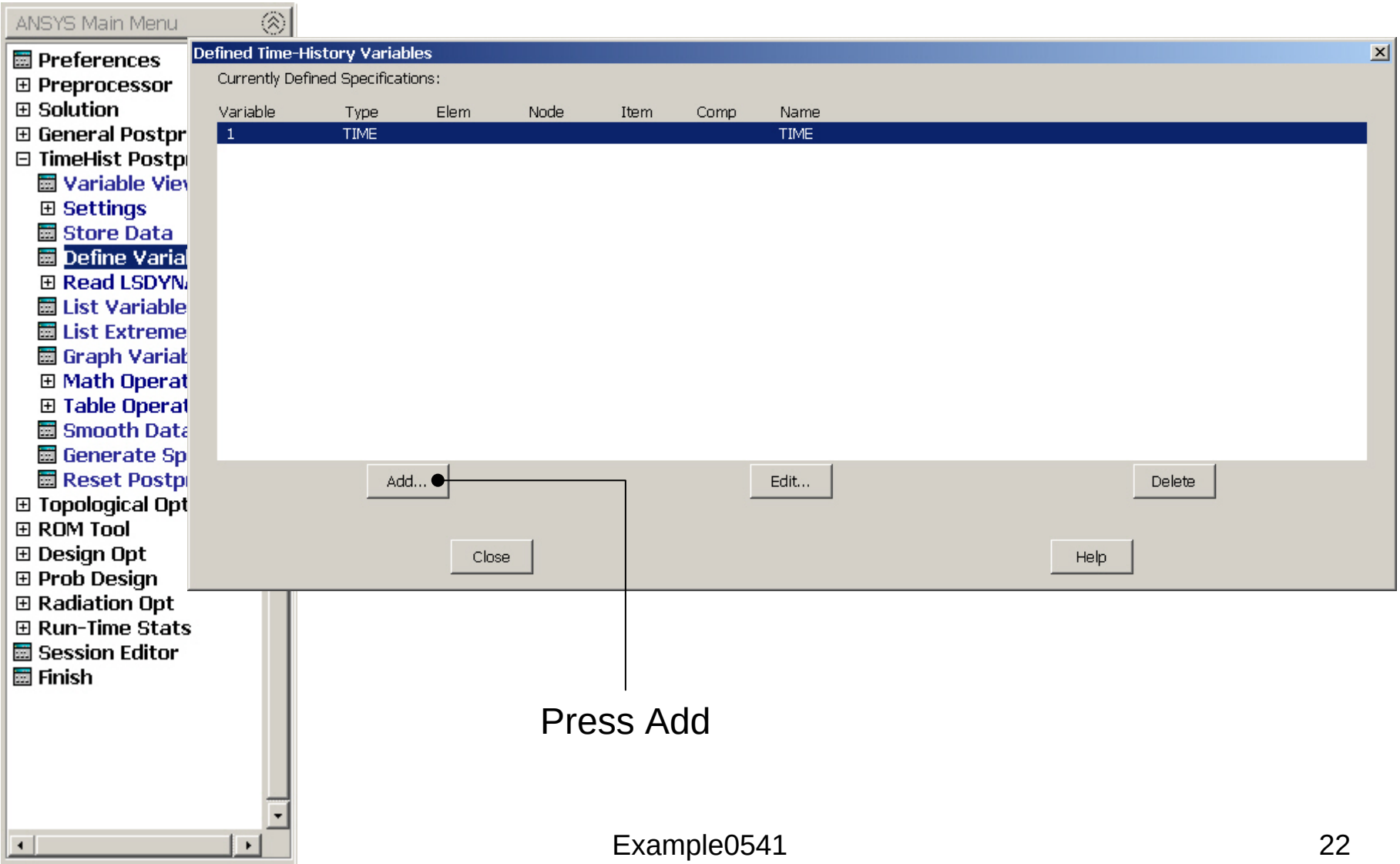


General Postproc > Plot Results > Deformed Shape



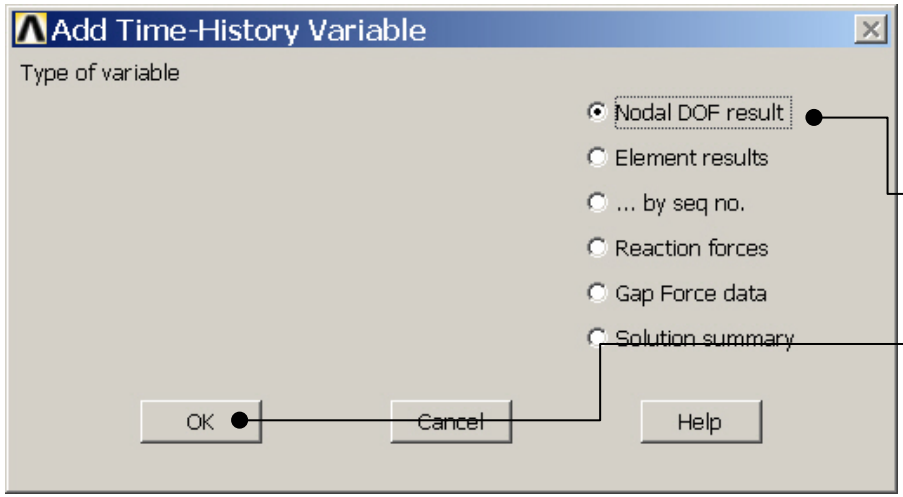
Select “Def+undeformed” and Press OK

Example – Define Variables



Press Add

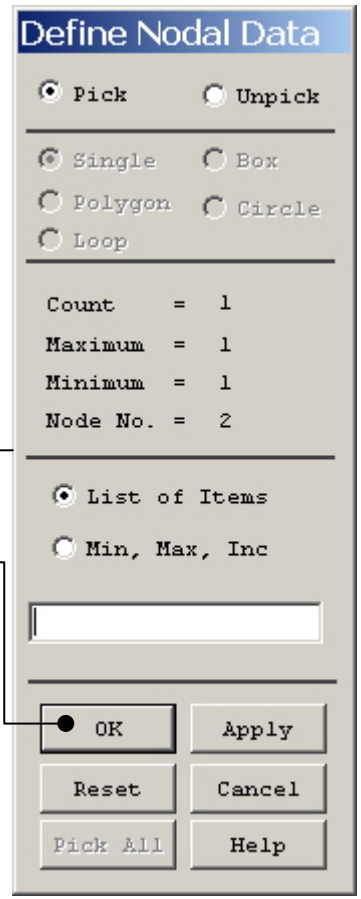
Example – Add Time-History Var.

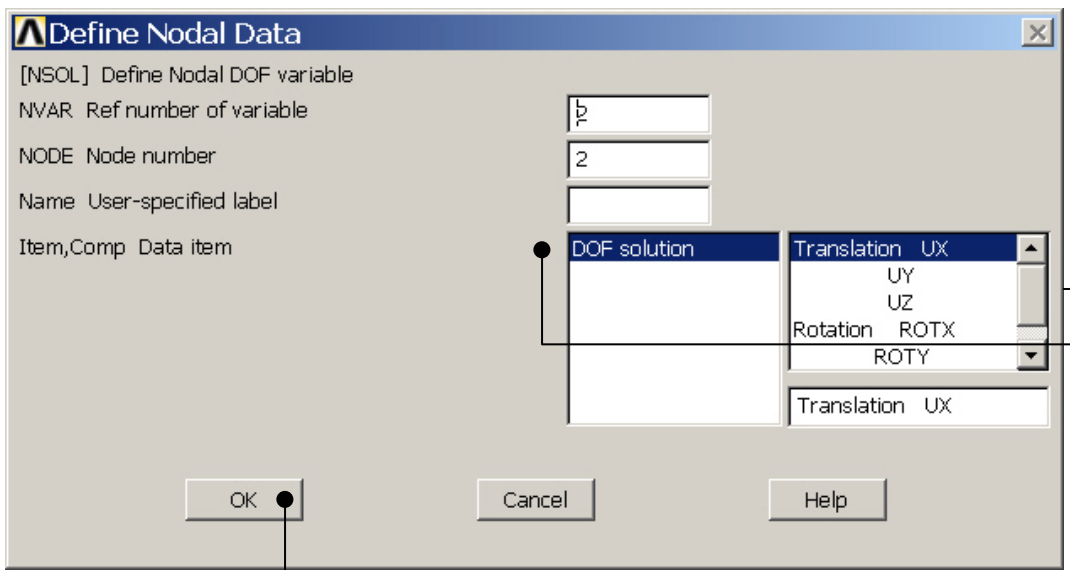


Pick the FZ node

Select Nodal DOF result

Press OK



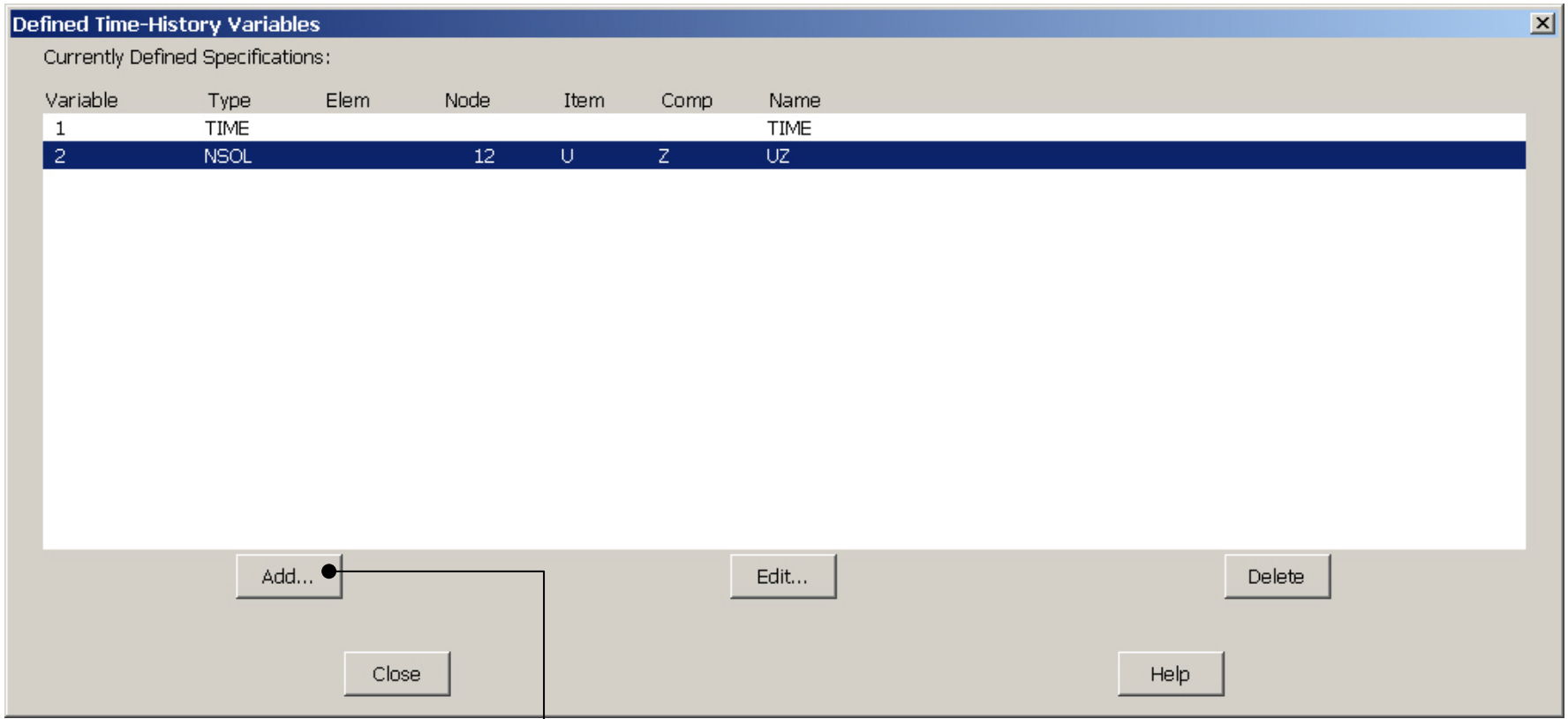


Press OK

Select DOF solution and Translation UZ

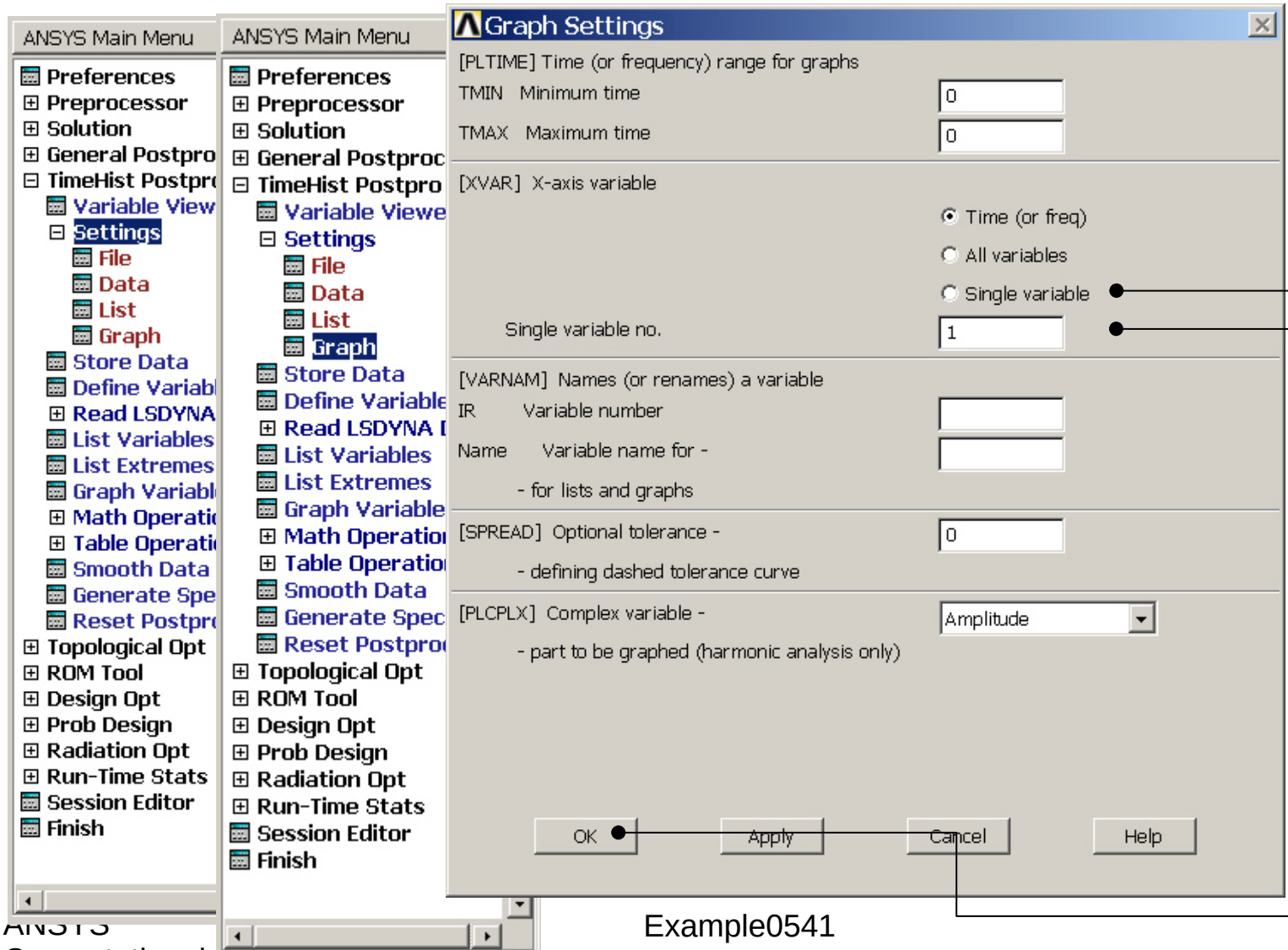
Press OK

Example – Add Time-History Var.



Press Add

Example - Settings

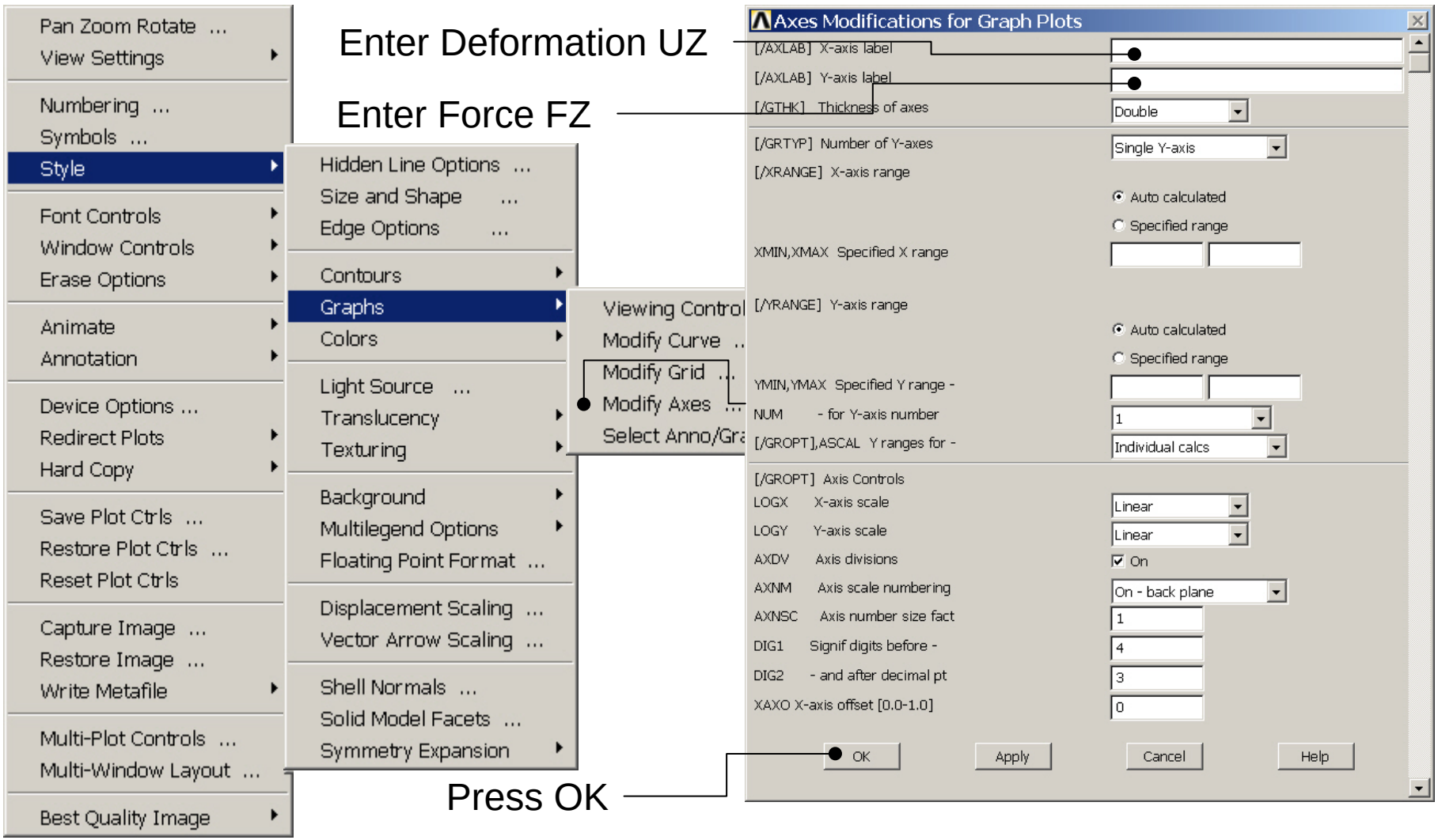


Select Single variable to plot on X-axis

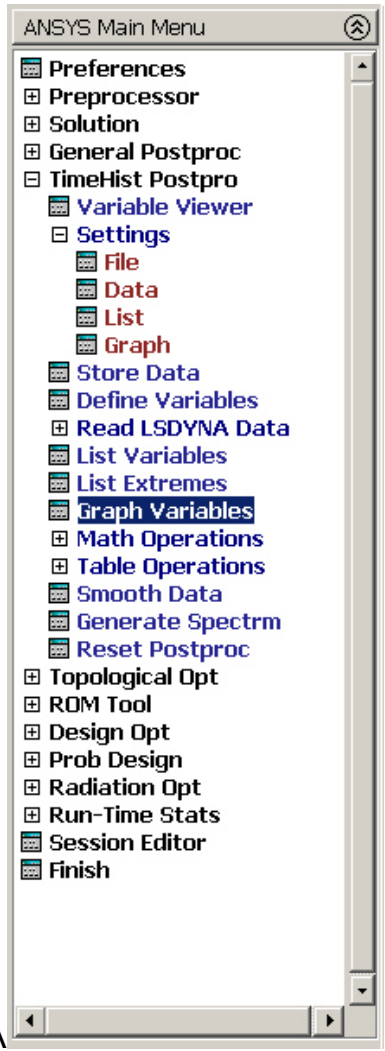
Enter 2 to plot UZ for the top node on the X-axis

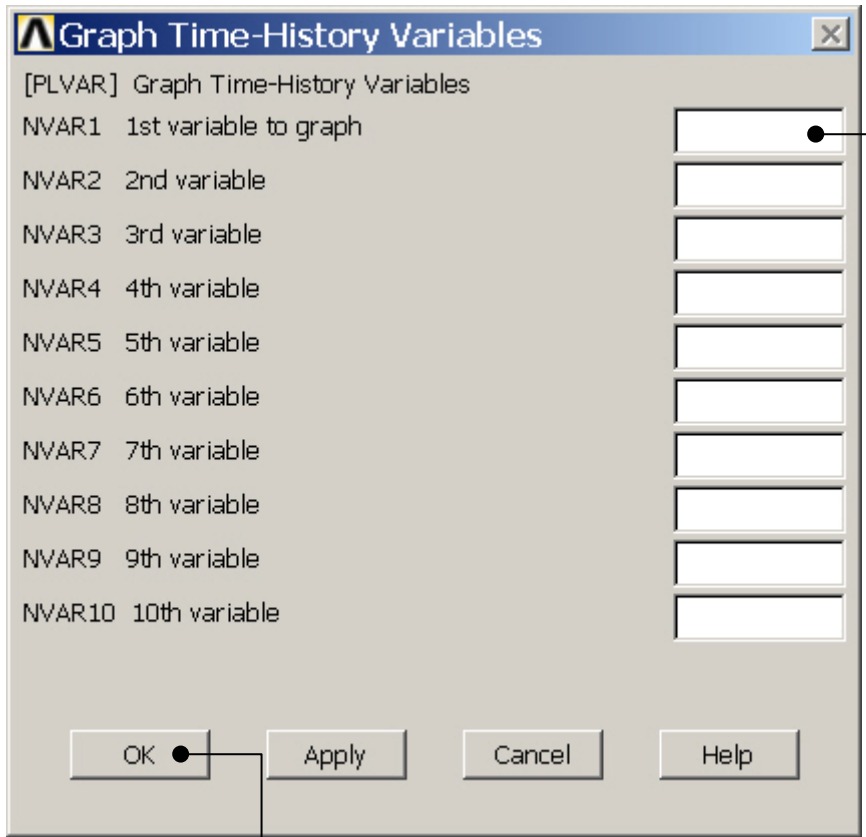
Press OK

Example – Style - Graph



Example – Graph Variables





Enter 1 to plot the reaction force FZ on the Y-axis

Press OK