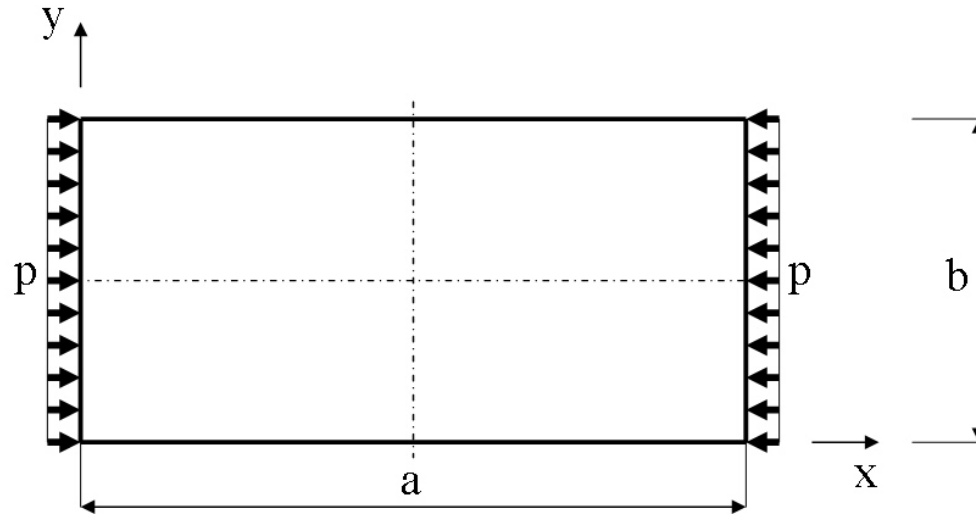


Course in ANSYS

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

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

$$\nu = 0.3$$

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

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

$$t = 1 \text{ mm}$$

$$p = 70 \text{ N/mm}$$

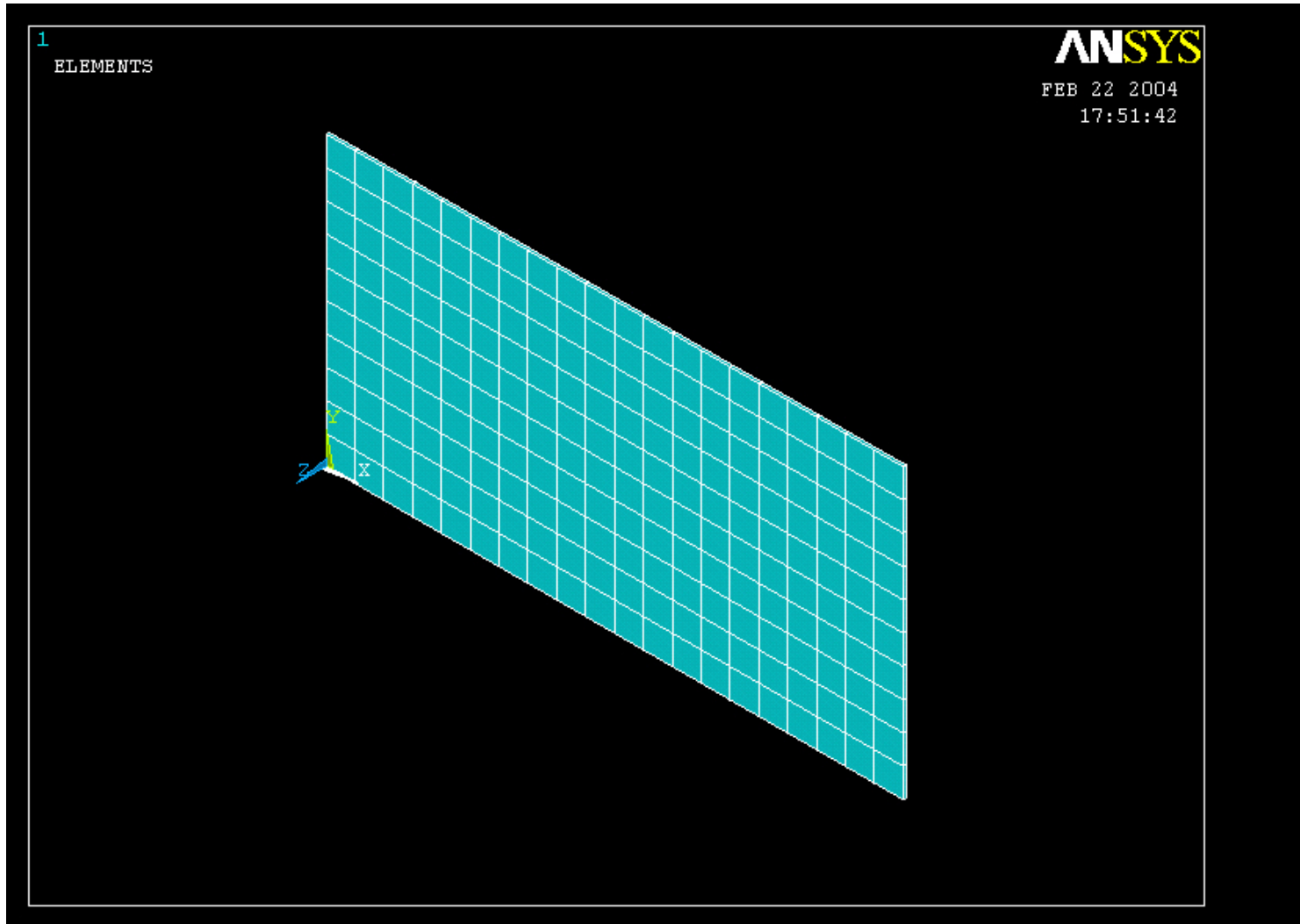
Example – Plate

- 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

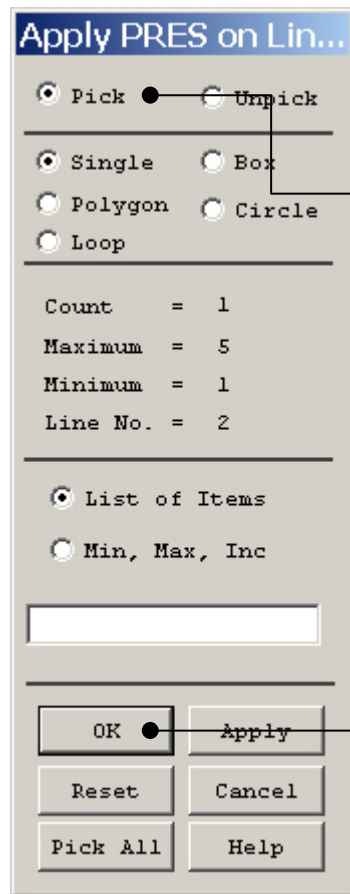
Load the example0505.lgw by **File Menu > Read input from**

Example - Plate



Example – Define Loads

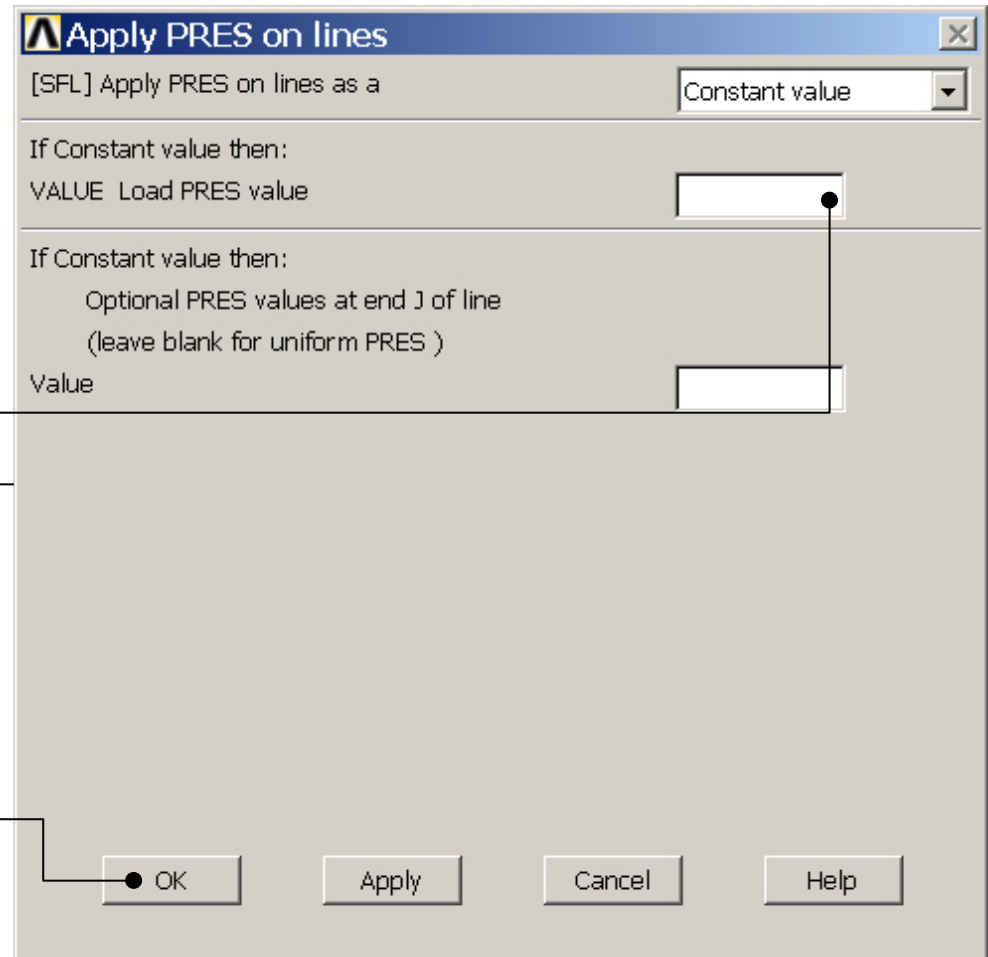
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the
left and
right
straight line

Enter 70

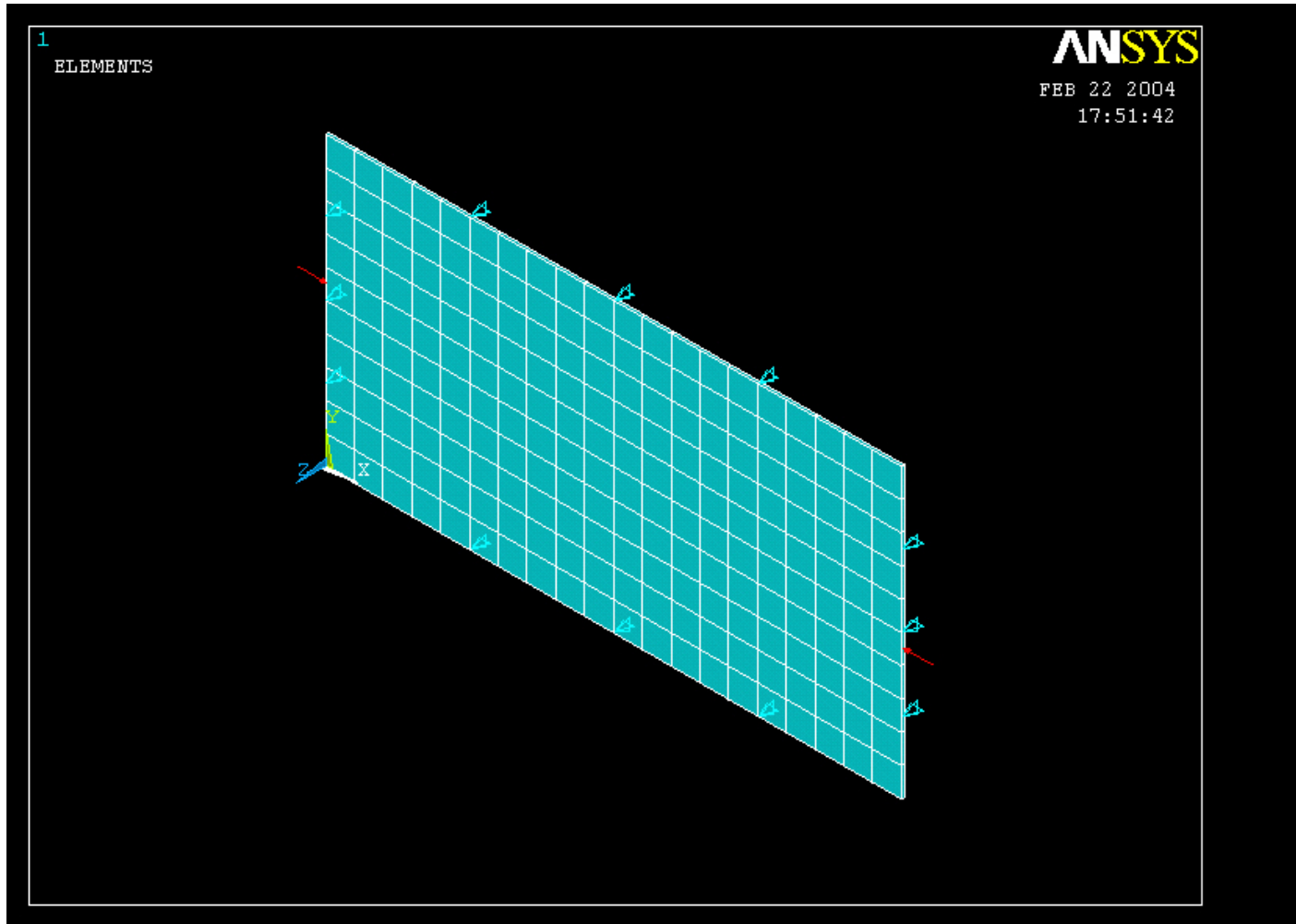
Press OK
to finish



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

Example0541

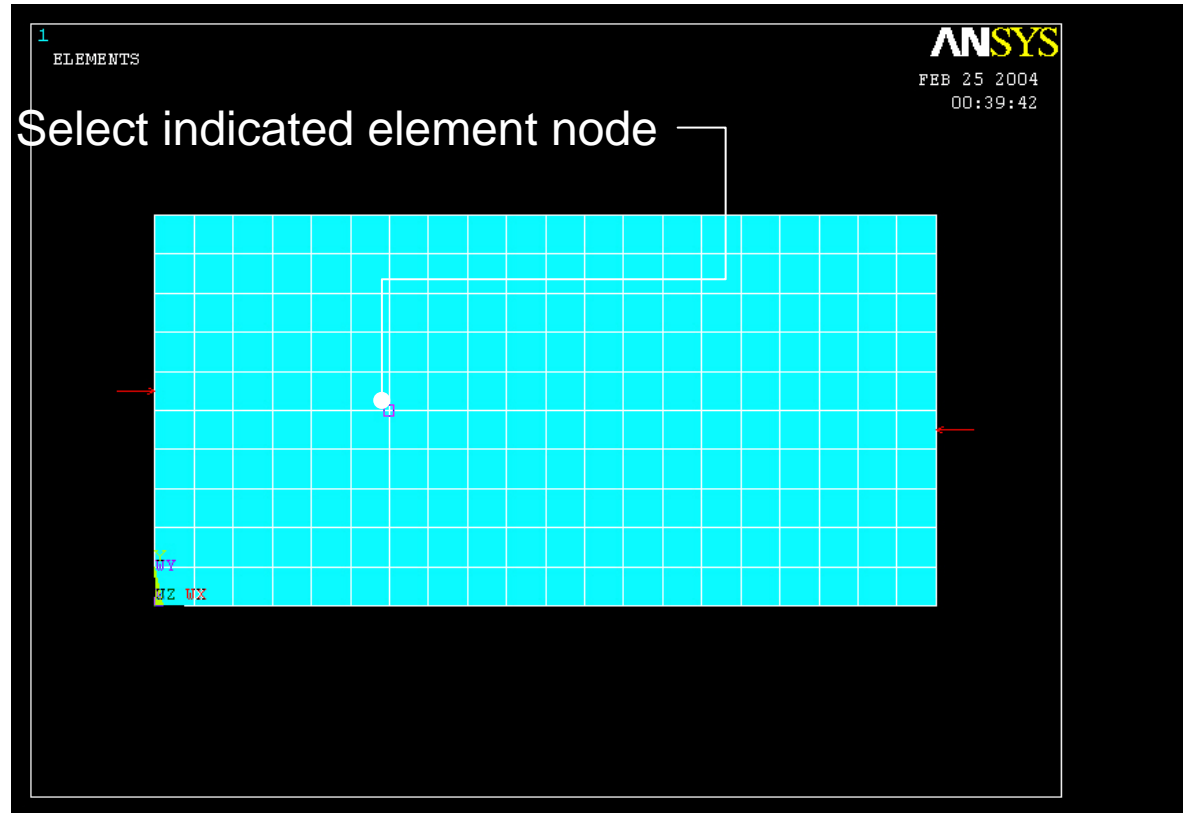
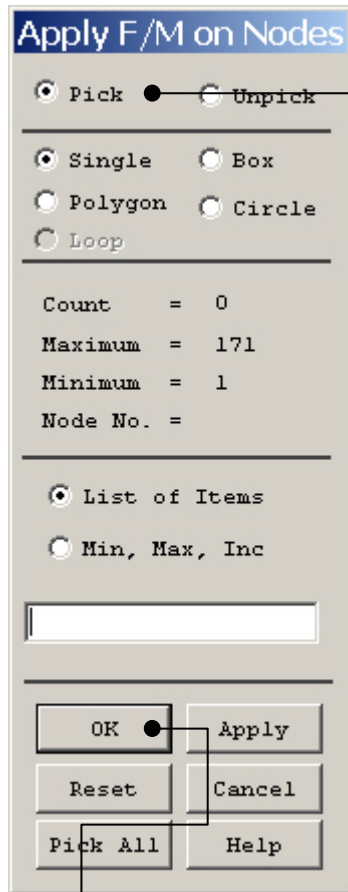
Example - Plate



Example – Define Loads

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

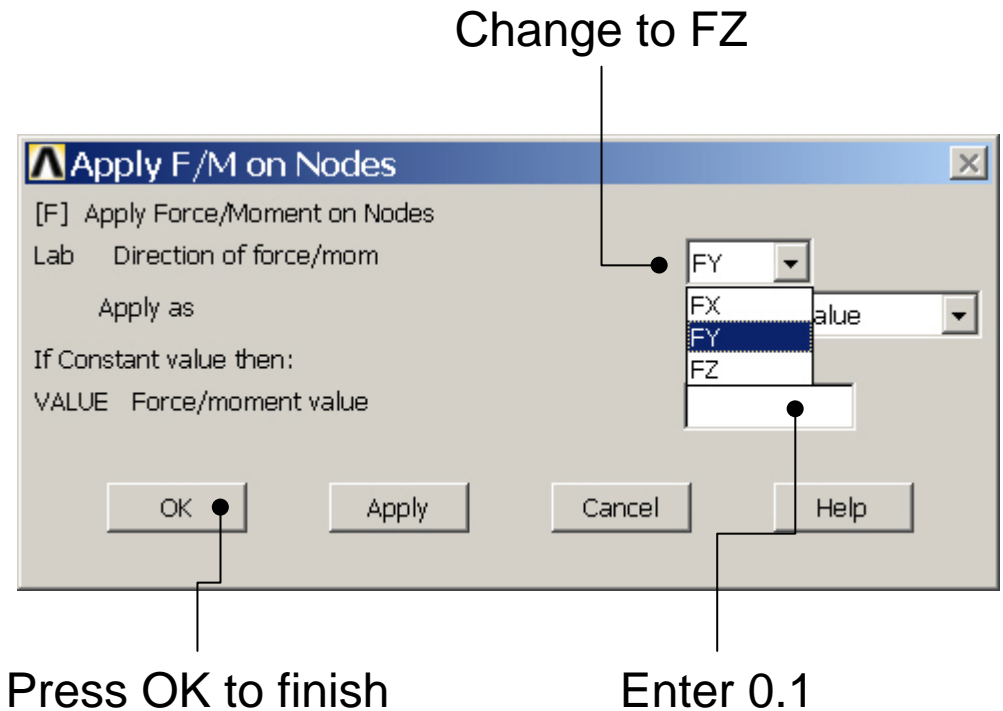
Note: If the model is remeshed all loads will be deleted with the element nodes



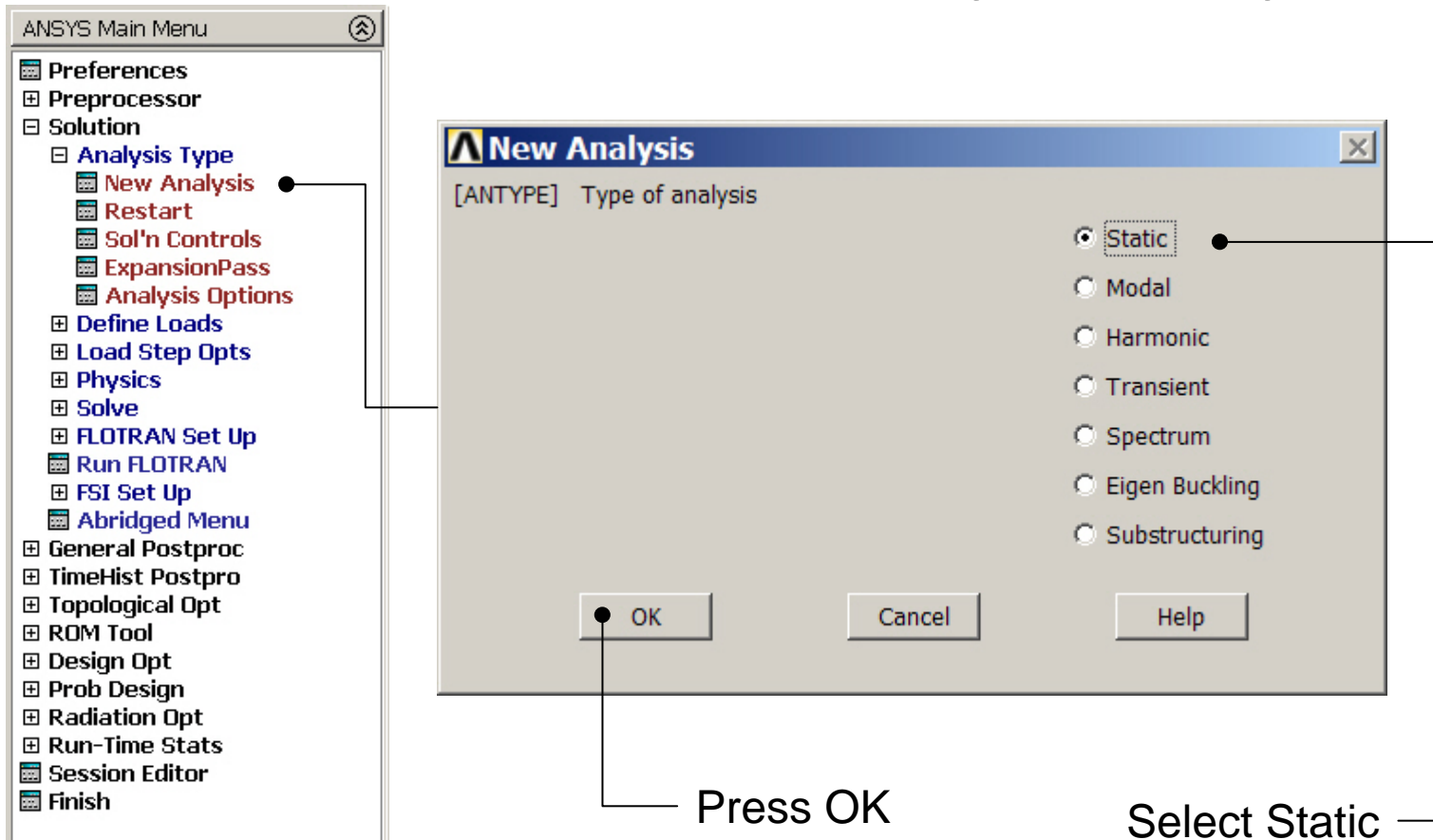
Press OK

Example – Define Loads

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

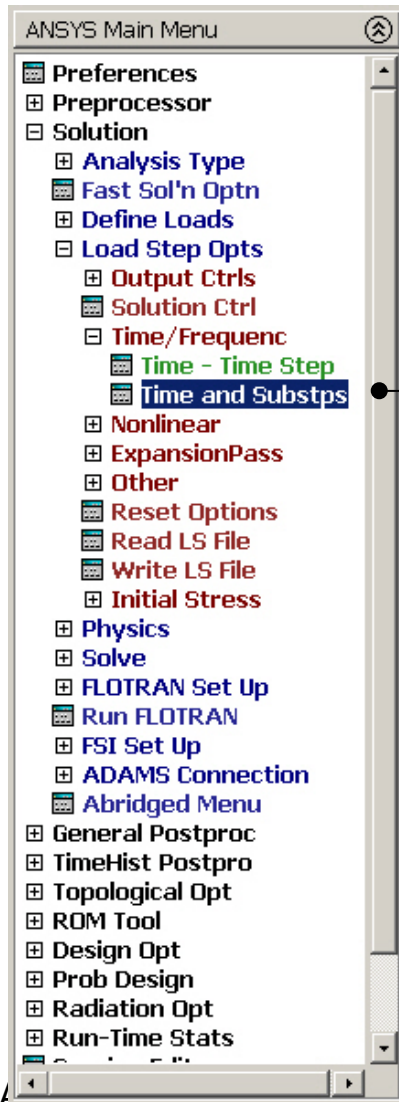


Example – Analysis Type



Example0541

Example – Load Step Opts



Time and Substep Options

Time and Substep Options

[TIME] Time at end of load step: 0

[NSUBST] Number of substeps:

[KBC] Stepped or ramped b.c.: ☒ Ramped ☐ Stepped

[AUTOTS] Automatic time stepping: ☐ ON ☐ OFF ☒ Prog Chosen

[NSUBST] Maximum no. of substeps:

Minimum no. of substeps:

Use previous step size? ☒ Yes

[TSRES] Time step reset based on specific time points

Time points from : ☒ No reset ☐ Existing array ☐ New array

Note: TSRES command is valid for thermal elements, thermal-electric elements, thermal surface effect elements and FLUID116, or any combination thereof.

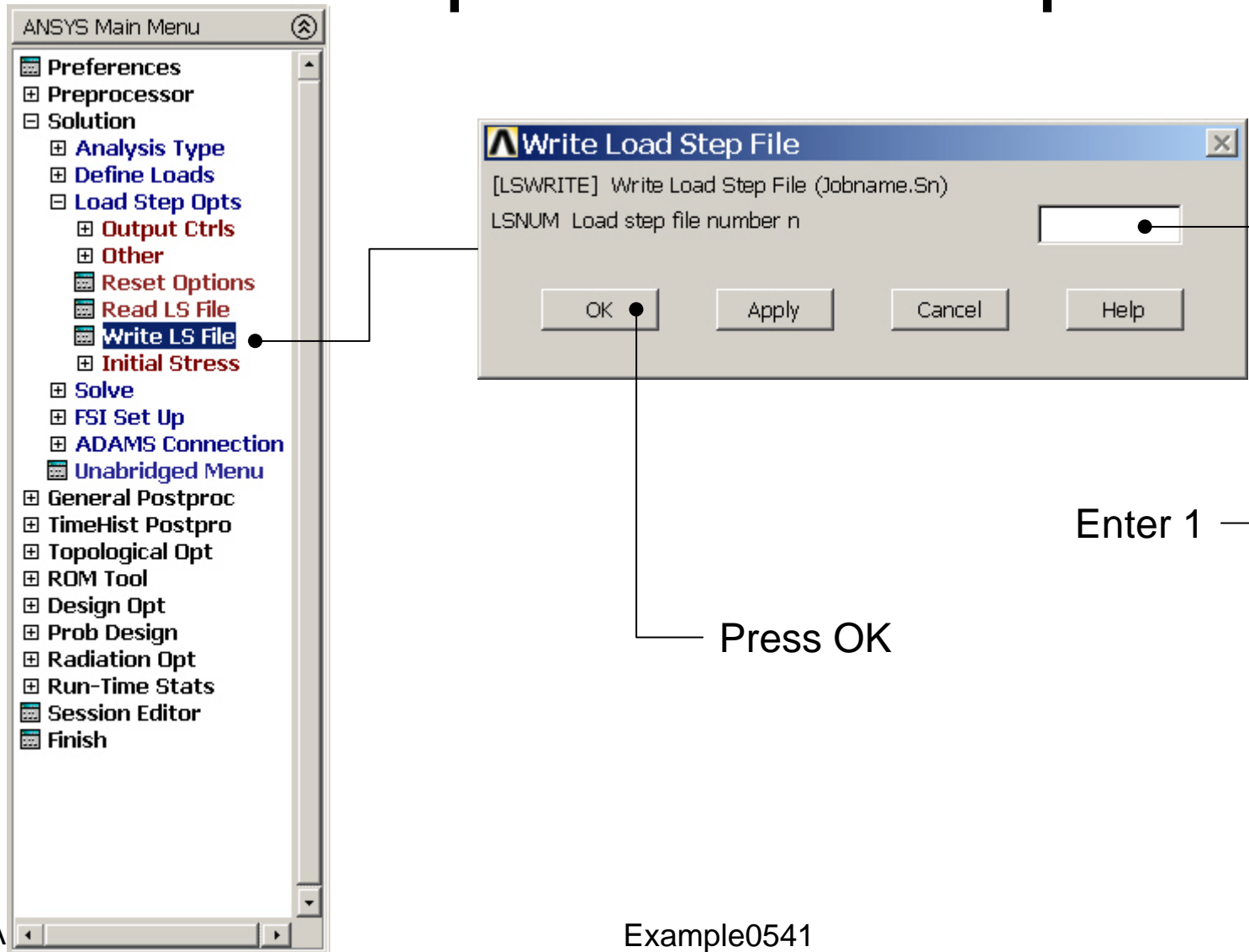
OK Cancel Help

Enter 70

Enter 3

Example0541

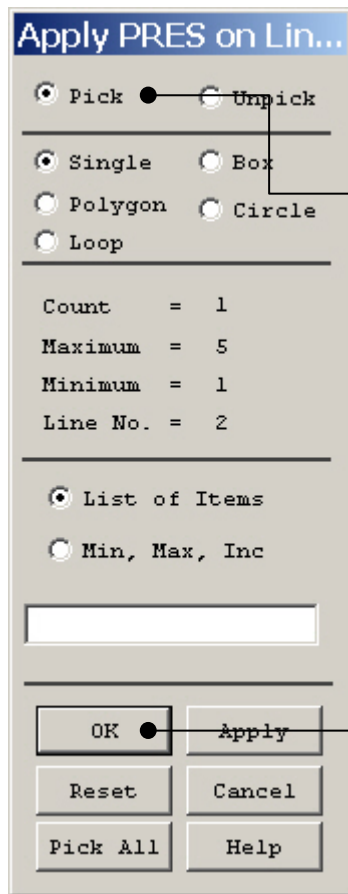
Example – Loadstep file



Example0541

Example – Define Loads

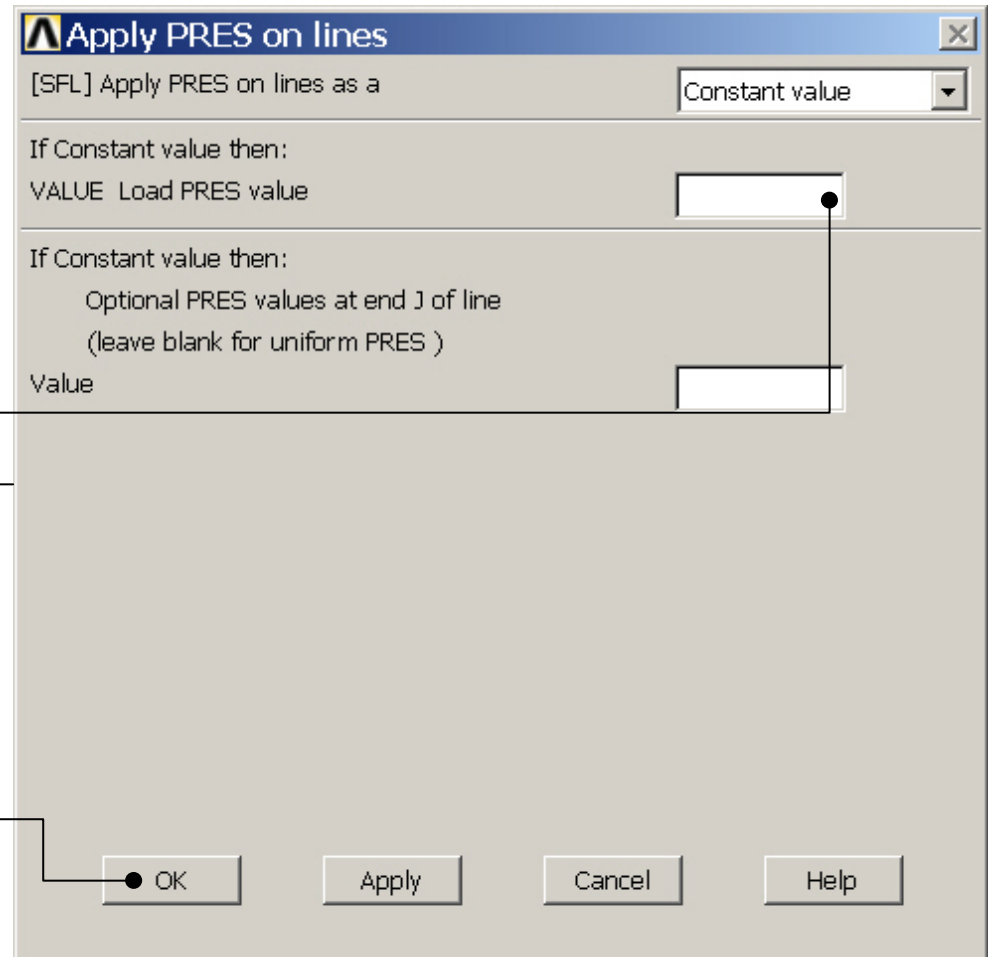
Solution > Define Loads > Apply > Structural > Pressure > On lines



Select the
left and
right
straight line

Enter 80

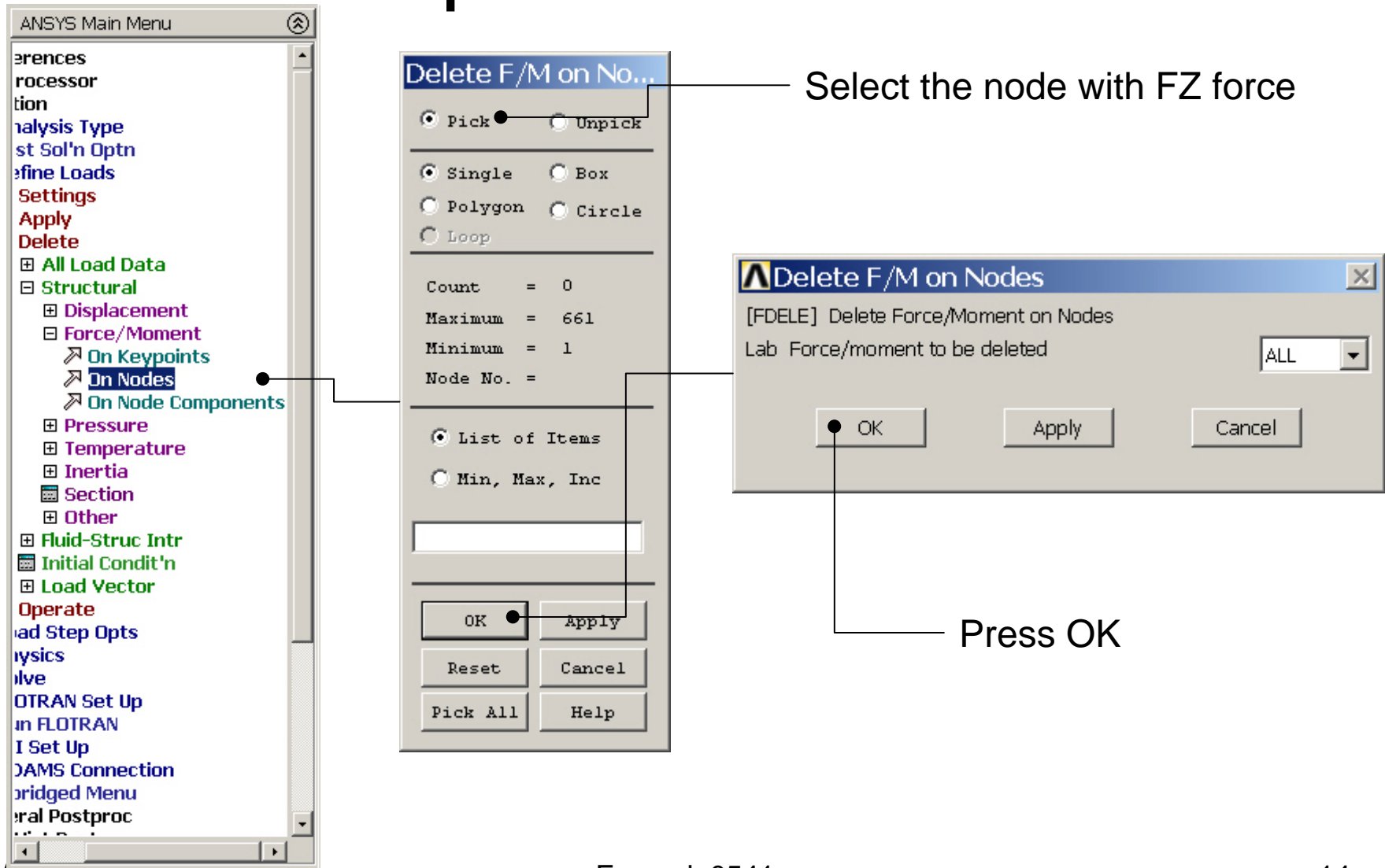
Press OK
to finish



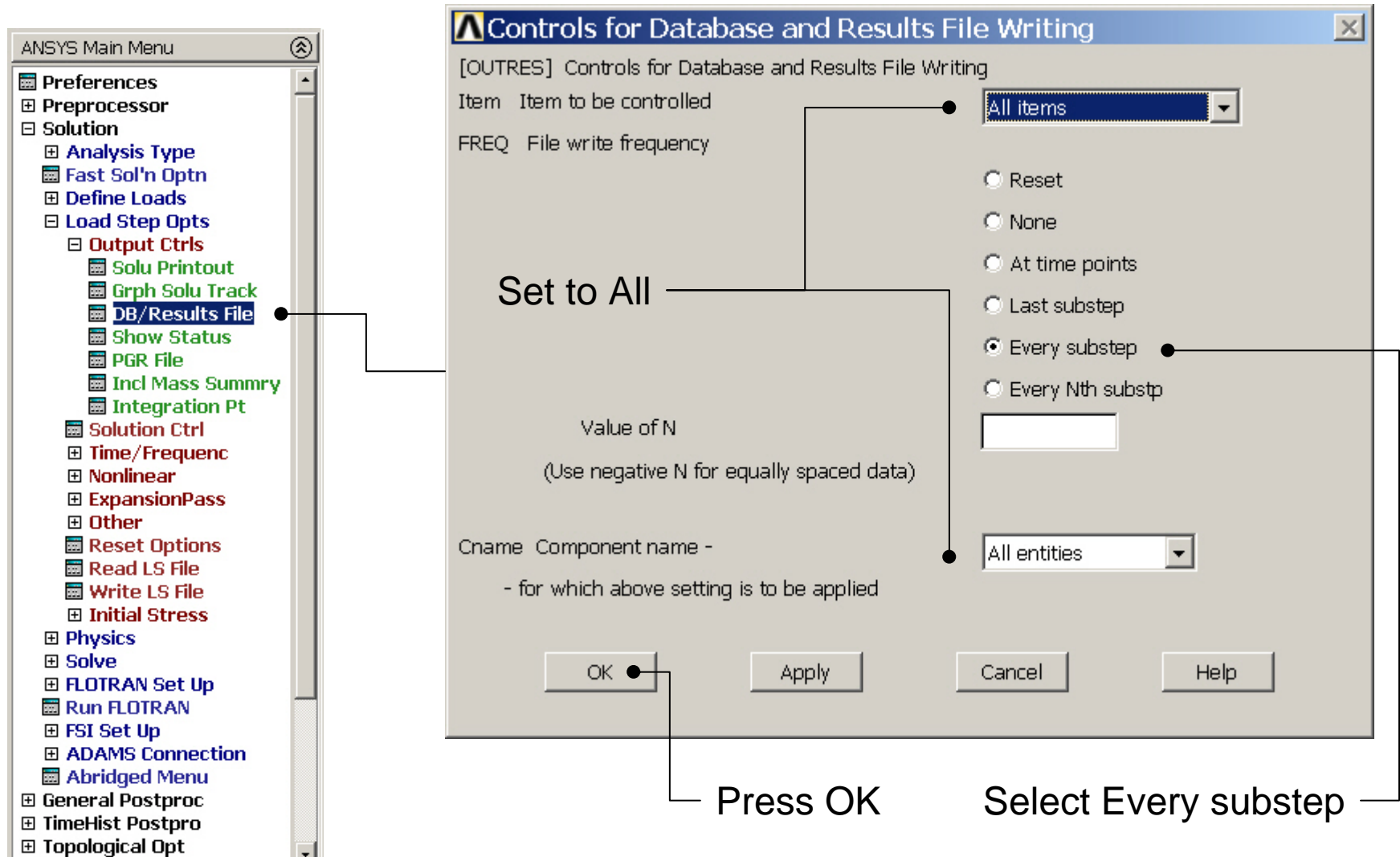
Example0541

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

Example – Delete Load



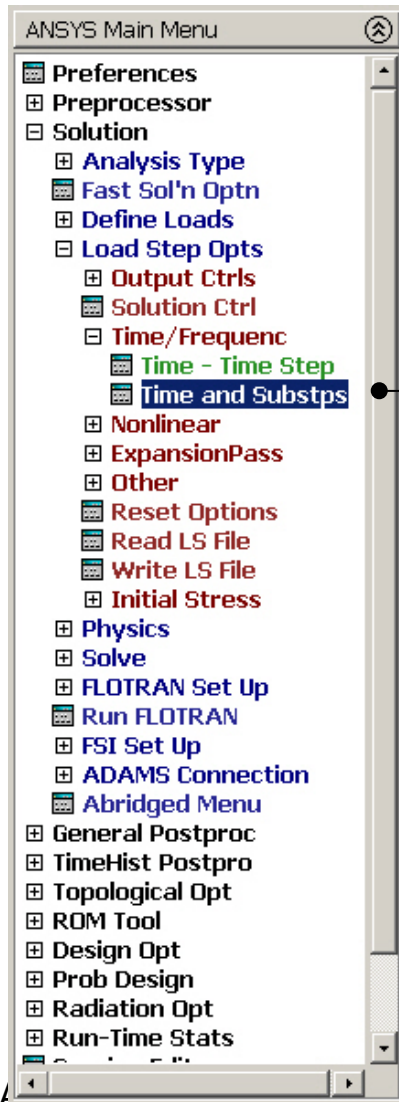
Example – Output Ctrls



Example0541

15

Example – Load Step Opts



Time and Substep Options

Time and Substep Options

[TIME] Time at end of load step: 0

[NSUBST] Number of substeps:

[KBC] Stepped or ramped b.c.: ☒ Ramped ☐ Stepped

[AUTOTS] Automatic time stepping: ☐ ON ☐ OFF ☒ Prog Chosen

[NSUBST] Maximum no. of substeps:

Minimum no. of substeps:

Use previous step size? ☒ Yes

[TSRES] Time step reset based on specific time points
Time points from : ☒ No reset ☐ Existing array ☐ New array

Note: TSRES command is valid for thermal elements, thermal-electric elements, thermal surface effect elements and FLUID116, or any combination thereof.

OK Cancel Help

Enter 80

Enter 10

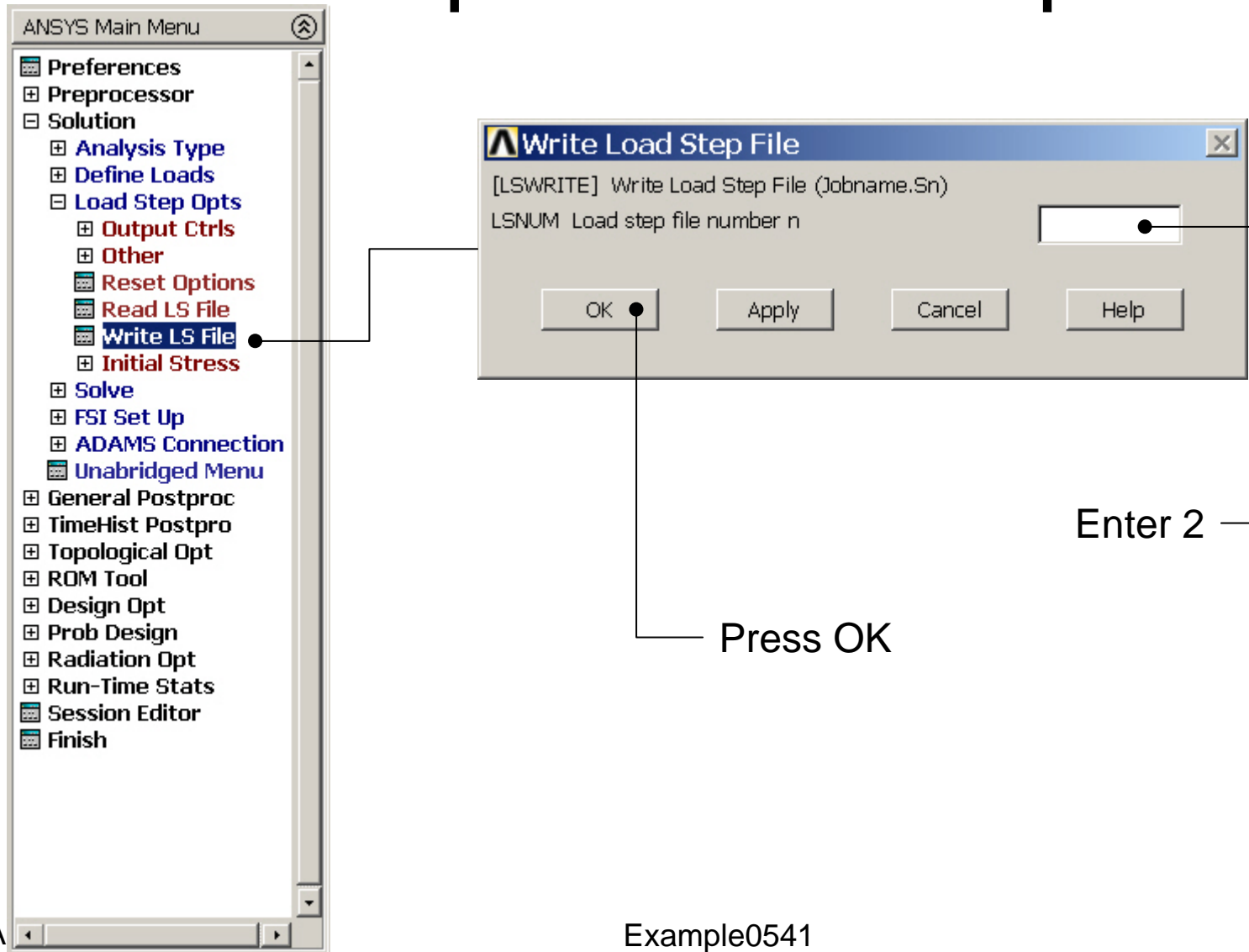
Enter 10

In case
ANSYS
decides to
perform
automatic
time
stepping

Example0541

16

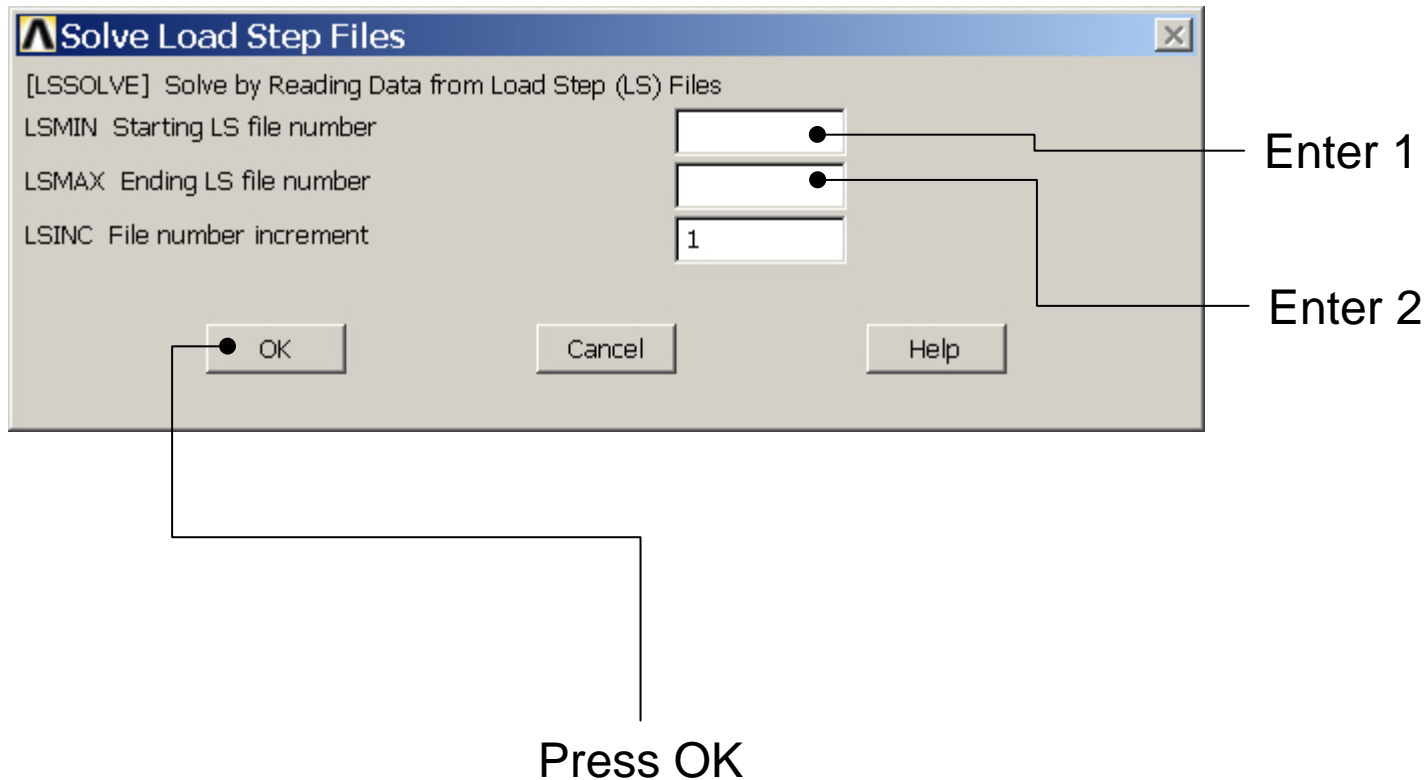
Example – Loadstep file



Example0541

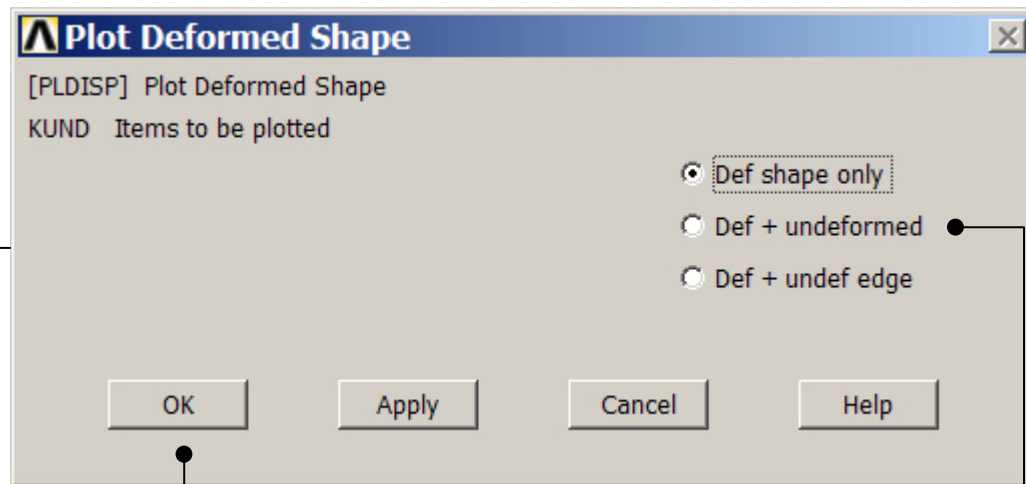
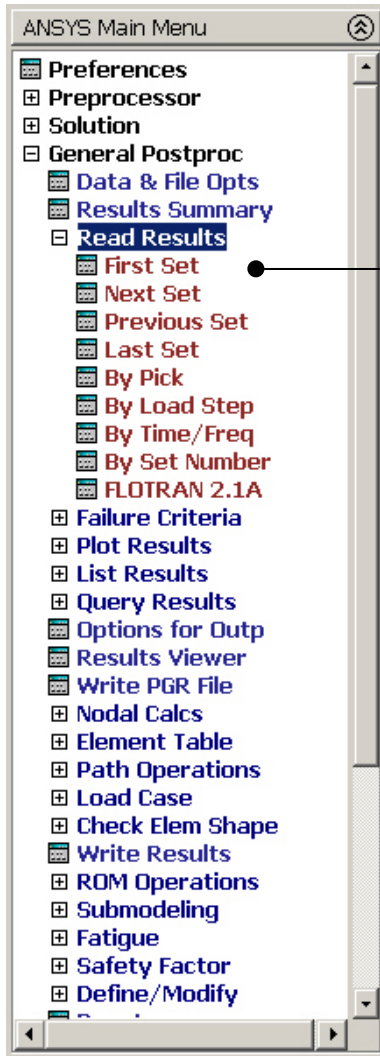
Example - Solve

Solution > Solve > From LS Files



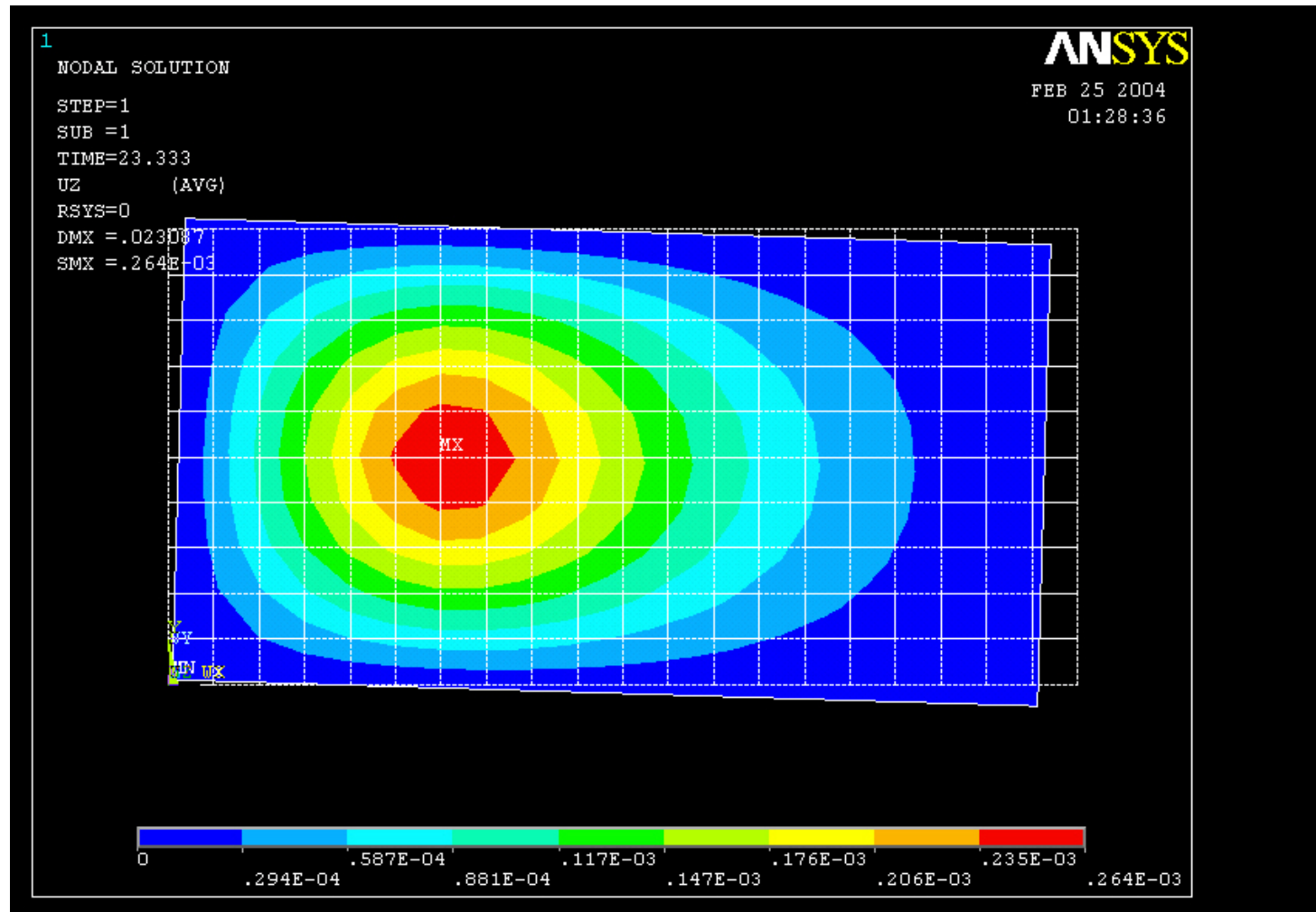
Example – Read Results

General Postproc > Plot Results > Deformed Shape



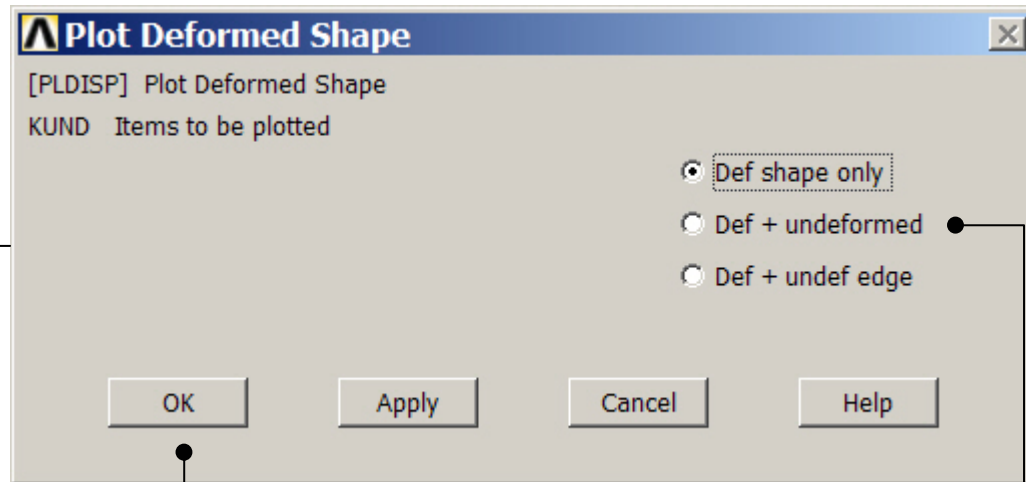
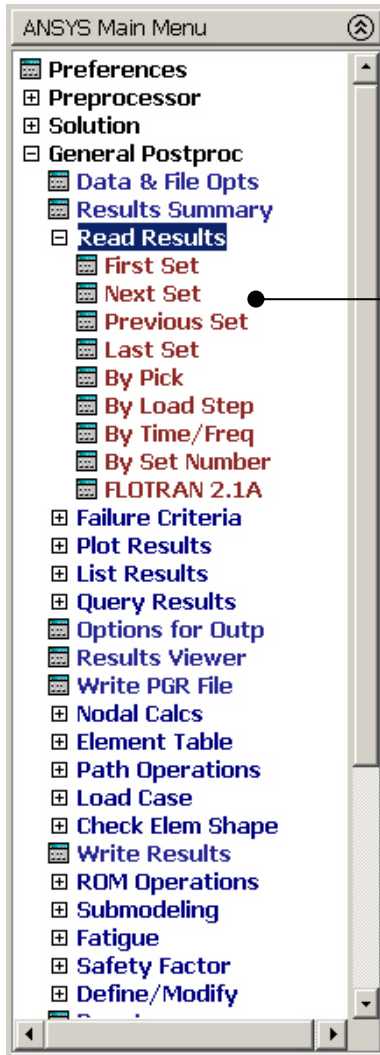
Select "Def+undeformed"
and Press OK

Example – Contour Plot



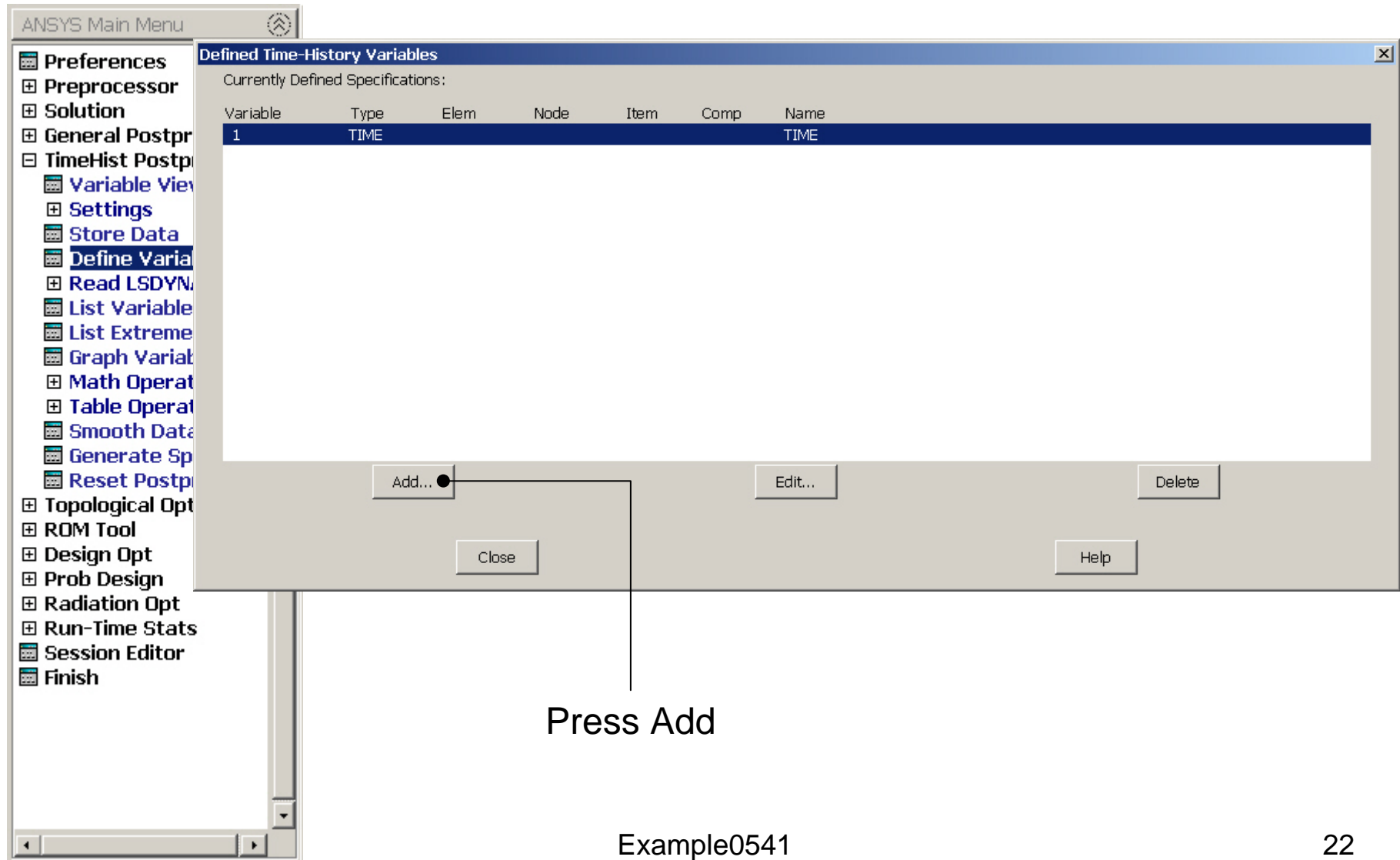
Example – Read Results

General Postproc > Plot Results > Deformed Shape

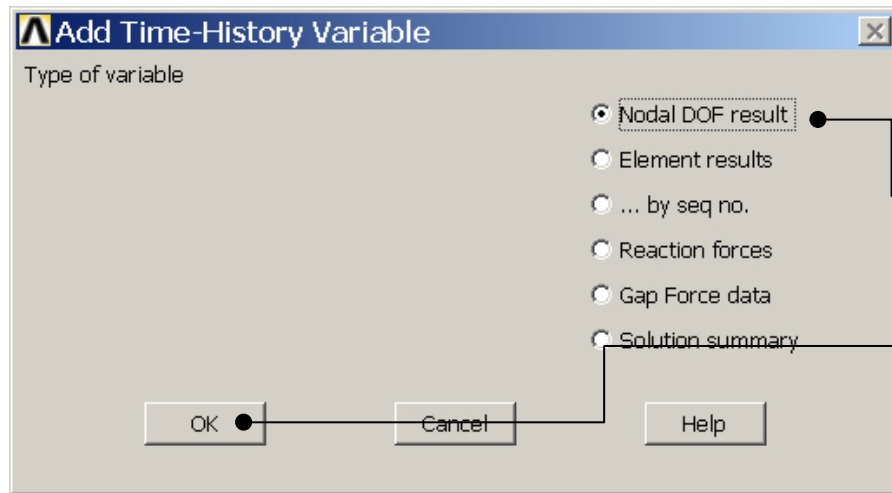


Select "Def+undeformed"
and Press OK

Example – Define Variables



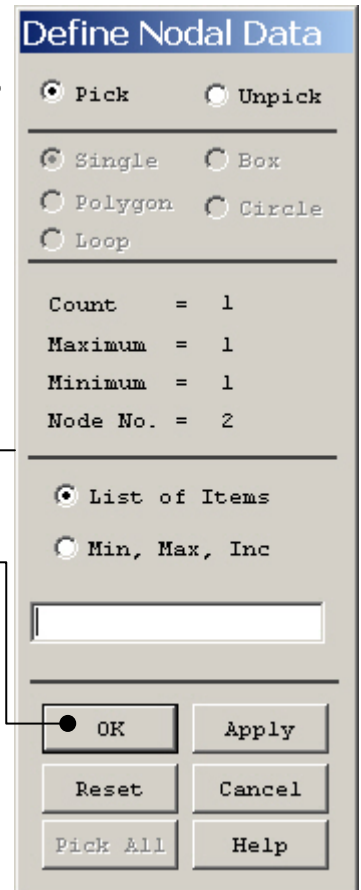
Example – Add Time-History Var.



Pick the FZ node

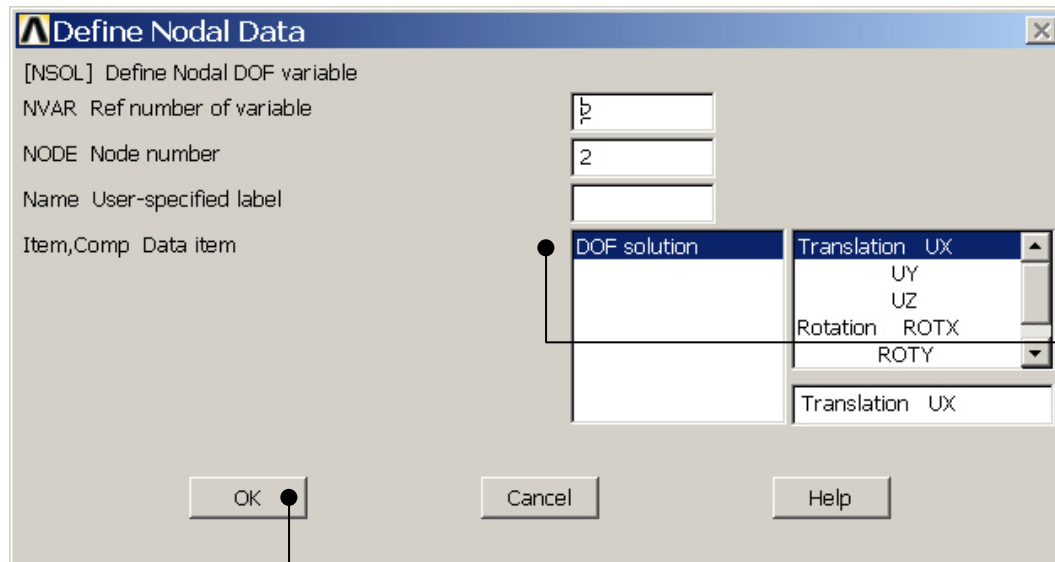
Select Nodal
DOF result

Press OK



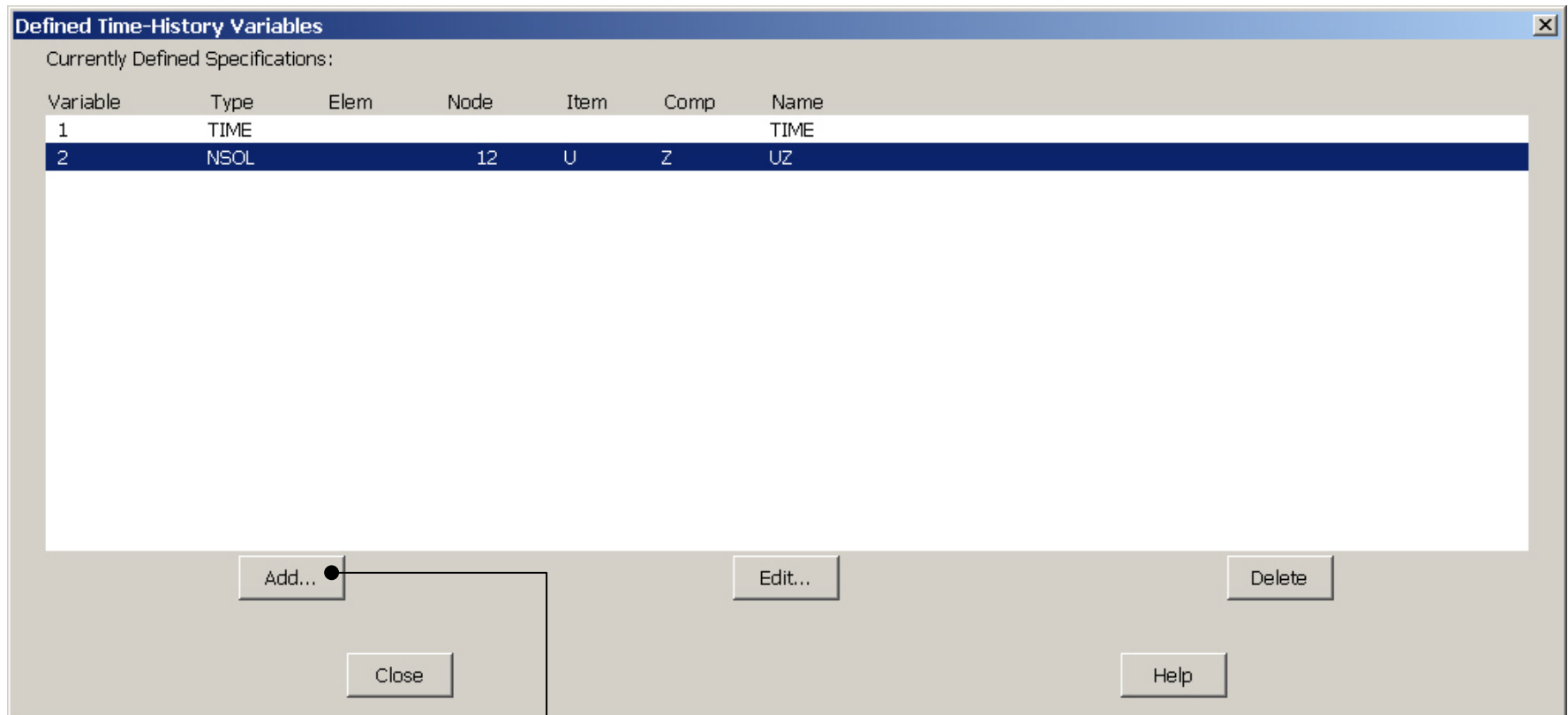
Press OK

Select DOF solution
and Translation UZ



Press OK

Example – Add Time-History Var.



Example - Settings

The image shows the ANSYS Main Menu on the left and the Graph Settings dialog box in the center. The Main Menu has two identical columns of options. The Graph Settings dialog box has the following fields and options:

- [PLTIME] Time (or frequency) range for graphs:
 - TMIN Minimum time: 0
 - TMAX Maximum time: 0
- [XVAR] X-axis variable:
 - ☒ Time (or freq)
 - ☐ All variables
 - ☐ Single variable
 - Single variable no.: 1
- [VARNAM] Names (or renames) a variable:
 - IR Variable number: [empty]
 - Name Variable name for - [empty]
 - for lists and graphs
- [SPREAD] Optional tolerance - 0
 - defining dashed tolerance curve
- [PLCPLX] Complex variable - Amplitude
 - part to be graphed (harmonic analysis only)

At the bottom of the dialog box are buttons for OK, Apply, Cancel, and Help.

Annotations on the right side of the image:

- Select Single variable to plot on X-axis (points to the Single variable radio button)
- Enter 2 to plot UZ for the top node on the X-axis (points to the Single variable no. field)
- Press OK (points to the OK button)

Example0541

25

Example – Style - Graph

The image shows the ANSYS software interface with the 'Style' menu open. The 'Style' menu is highlighted, and its sub-menu is visible. The 'Graphs' option is selected in the sub-menu. The 'Axes Modifications for Graph Plots' dialog box is open, showing various settings for graph axes. The 'OK' button is highlighted.

Menu Path:

- Pan Zoom Rotate ...
- View Settings
- Numbering ...
- Symbols ...
- Style**
- Font Controls
- Window Controls
- Erase Options
- Animate
- Annotation
- Device Options ...
- Redirect Plots
- Hard Copy
- Save Plot Ctrls ...
- Restore Plot Ctrls ...
- Reset Plot Ctrls
- Capture Image ...
- Restore Image ...
- Write Metafile
- Multi-Plot Controls ...
- Multi-Window Layout ...
- Best Quality Image

Style Sub-menu:

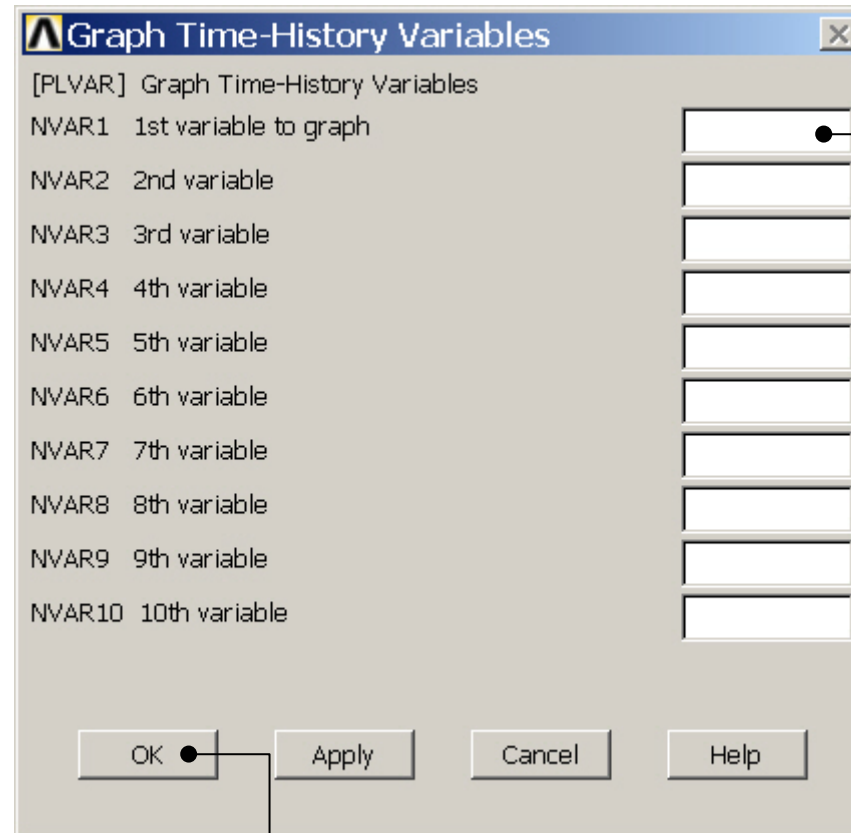
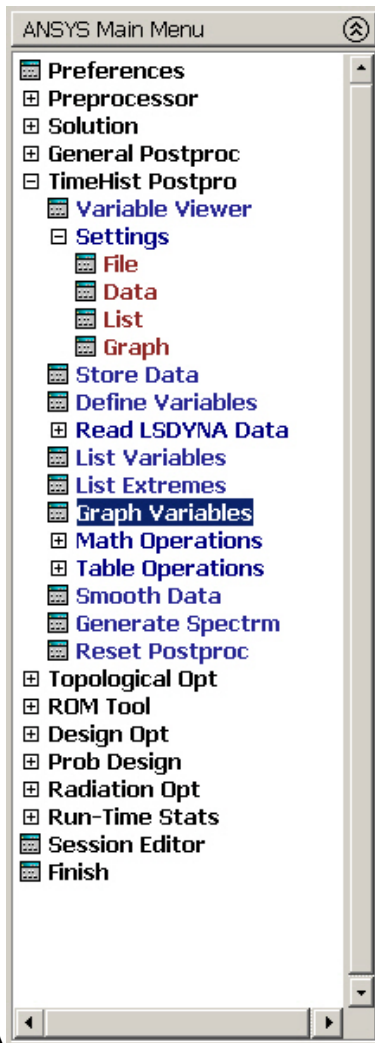
- Hidden Line Options ...
- Size and Shape ...
- Edge Options ...
- Contours
- Graphs**
- Colors
- Light Source ...
- Translucency
- Texturing
- Background
- Multilegend Options
- Floating Point Format ...
- Displacement Scaling ...
- Vector Arrow Scaling ...
- Shell Normals ...
- Solid Model Facets ...
- Symmetry Expansion

Axes Modifications for Graph Plots Dialog:

- [/AXLAB] X-axis label
- [/AXLAB] Y-axis label
- [/GTHK] Thickness of axes: Double
- [/GRTYP] Number of Y-axes: Single Y-axis
- [/XRange] X-axis range: Auto calculated
- XMIN,XMAX Specified X range
- [/YRange] Y-axis range: Auto calculated
- YMIN,YMAX Specified Y range -
- NUM - for Y-axis number: 1
- [/GROPT],ASCAL Y ranges for -
- [/GROPT] Axis Controls
- LOGX X-axis scale: Linear
- LOGY Y-axis scale: Linear
- AXDV Axis divisions: On
- AXNM Axis scale numbering: On - back plane
- AXNSC Axis number size fact: 1
- DIG1 Signif digits before -
- DIG2 - and after decimal pt: 3
- XAXO X-axis offset [0.0-1.0]: 0

Buttons: OK, Apply, Cancel, Help

Example – Graph Variables



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

Press OK

Example0541