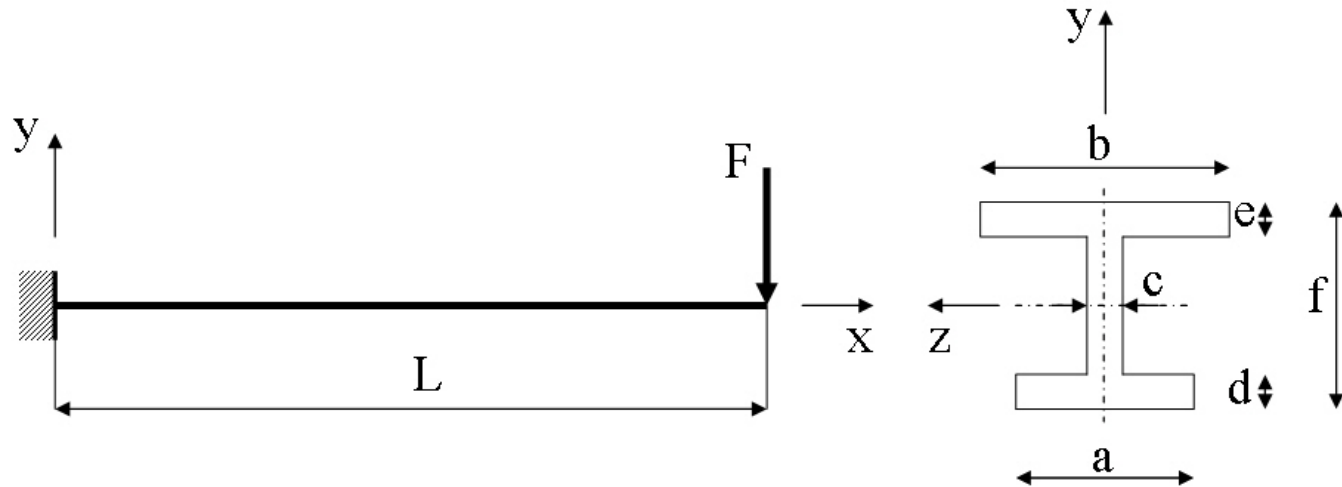


Course in ANSYS

Example0511

Example – Cantilever beam



Objective:

Plot the P-U curve for the nonlinear behaviour

Tasks:

Obtain a static solution including prestress

Obtain a buckling solution

Include imperfections using Update Geometry

Run the nonlinear analysis

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

$$\nu = 0.3$$

$$L = 5000 \text{ mm}$$

$$a = 250 \text{ mm}$$

$$b = 450 \text{ mm}$$

$$c = 10 \text{ mm}$$

$$d = 20 \text{ mm}$$

$$e = 15 \text{ mm}$$

$$f = 350 \text{ mm}$$

$$F = ?$$

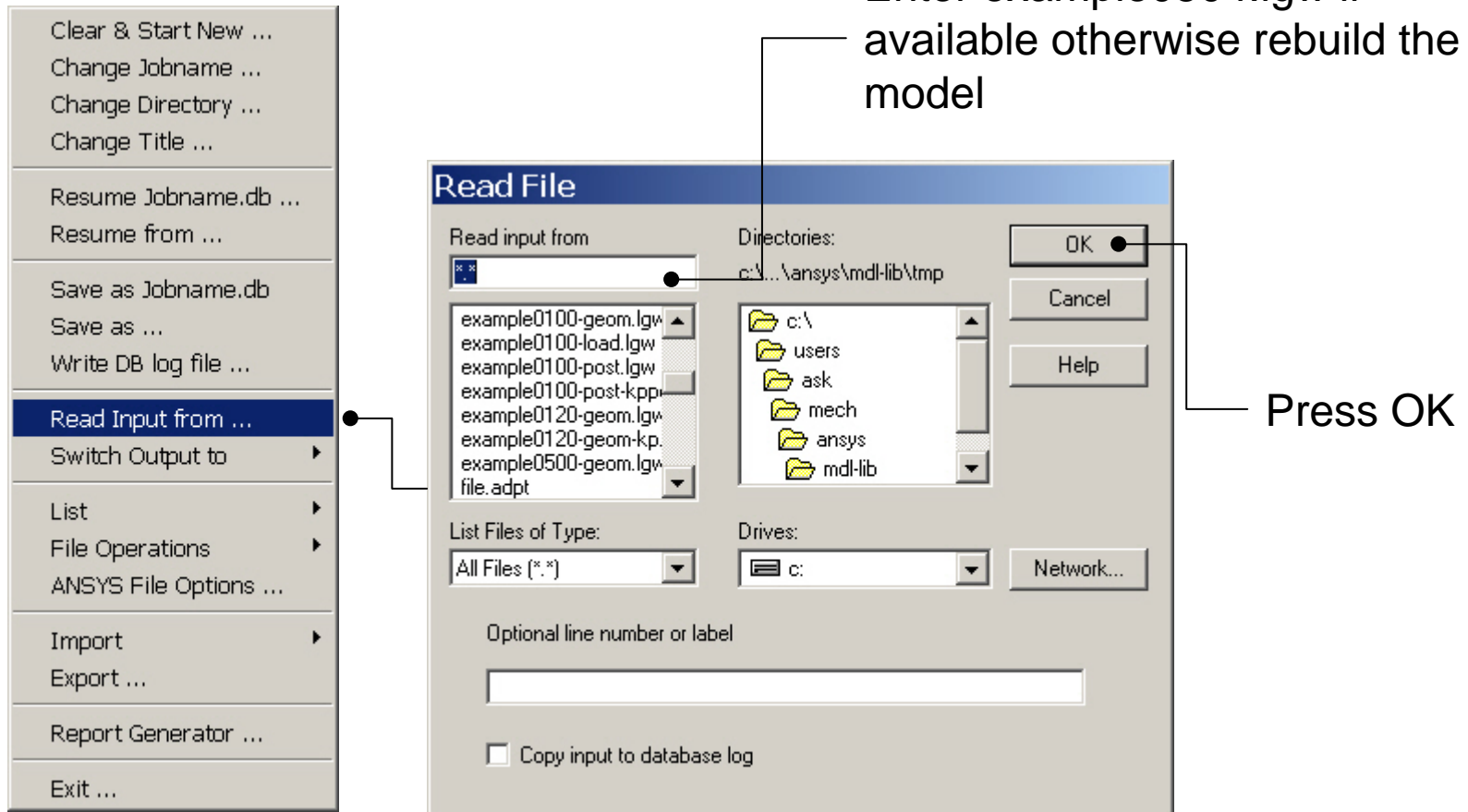
Nonlinear - Solution Phases

- Tasks
 - Run a static analysis with Prestress ON
 - Run a Eigen Buckling analysis with a unit load
 - ExpansionPass ON
 - Save the model
 - Finish the Solution process
 - – Plot results
 - Update geometry for a relevant buckling mode
 - In place of the unit load apply a load with a magnitude of the buckling load found for a relevant buckling mode
 - Run a static nonlinear analysis
 - Plot appropriate deformations vs. forces

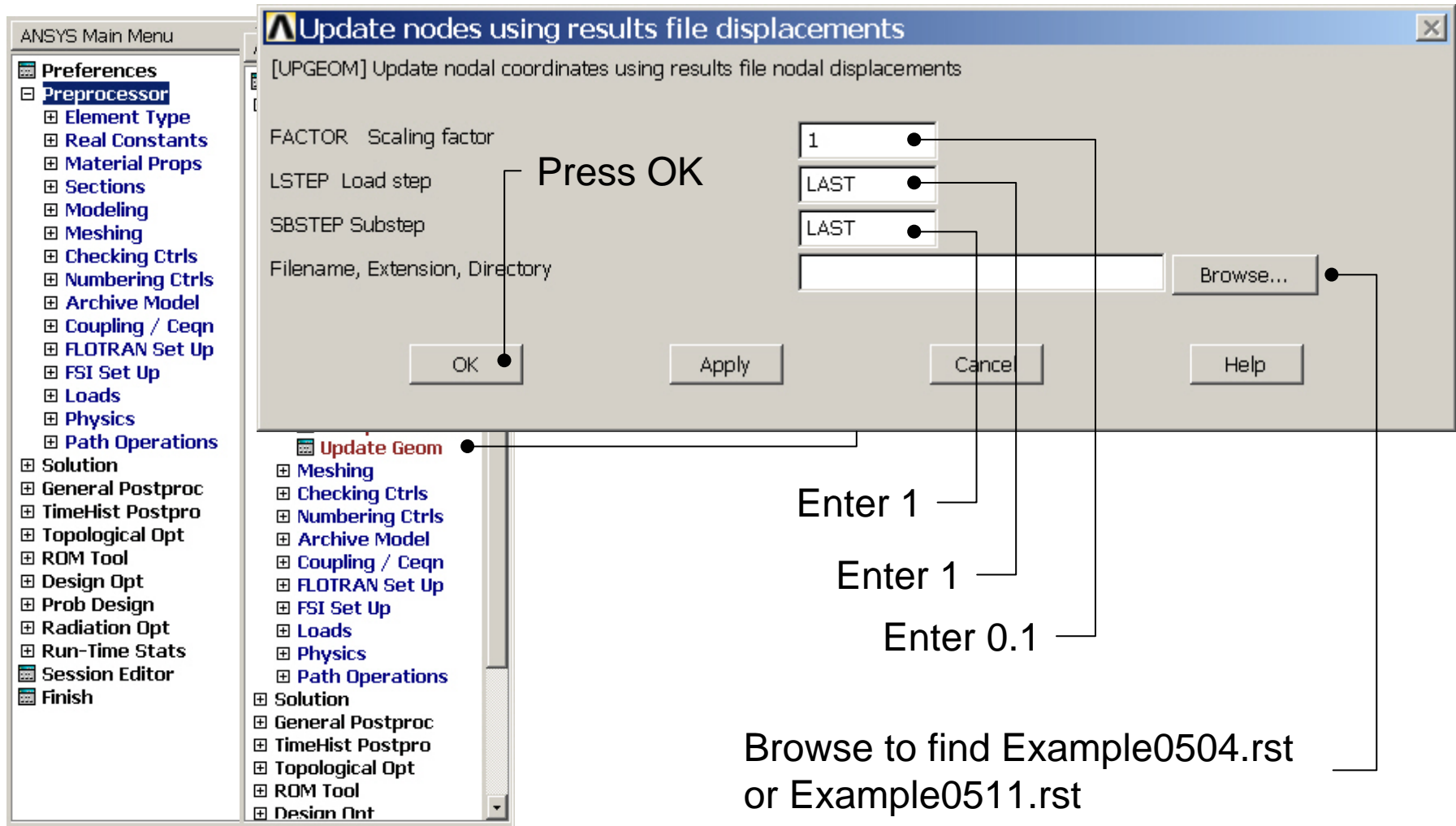
We start
here



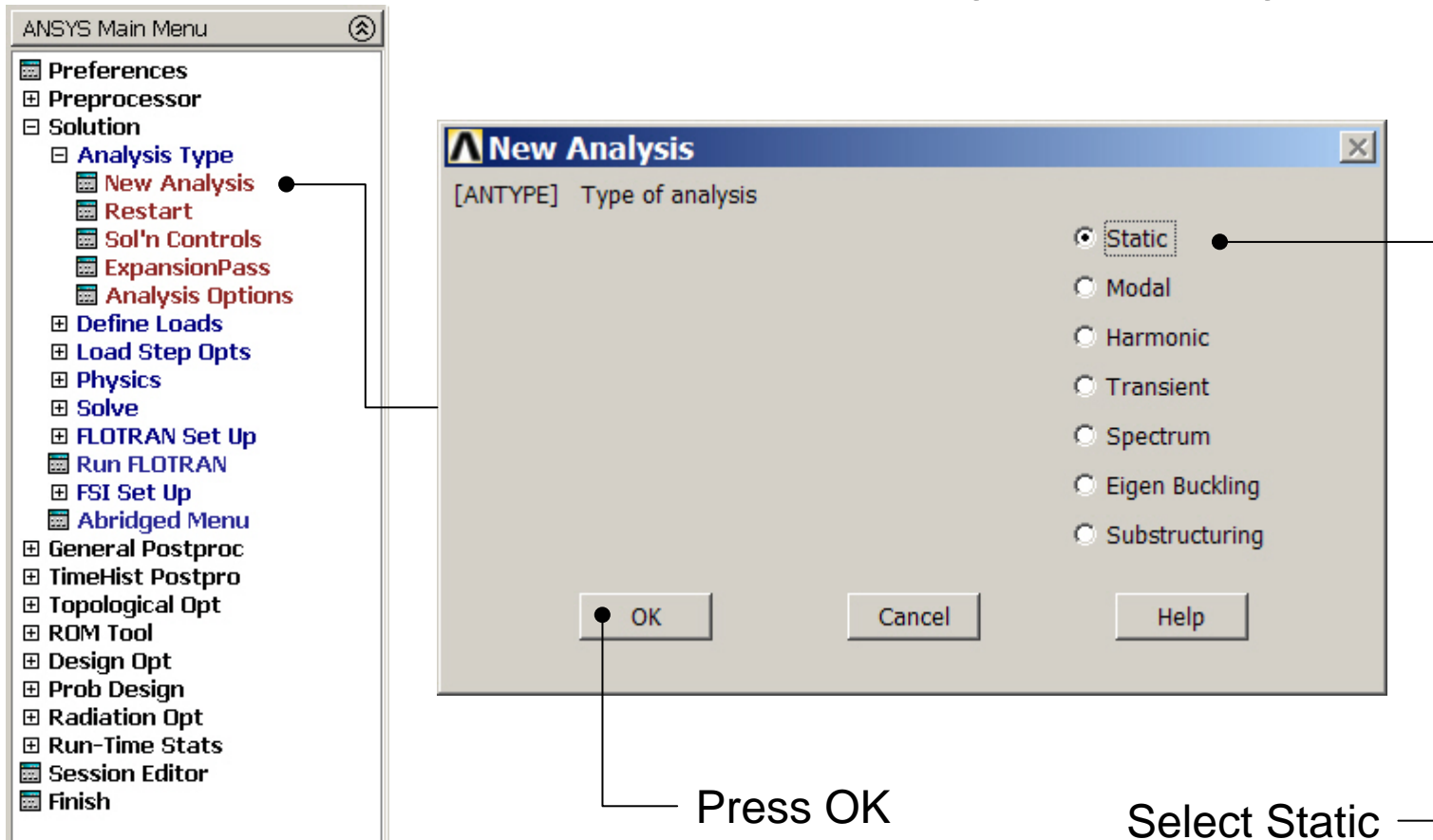
Example – Read Input from..



Example - Update Geom



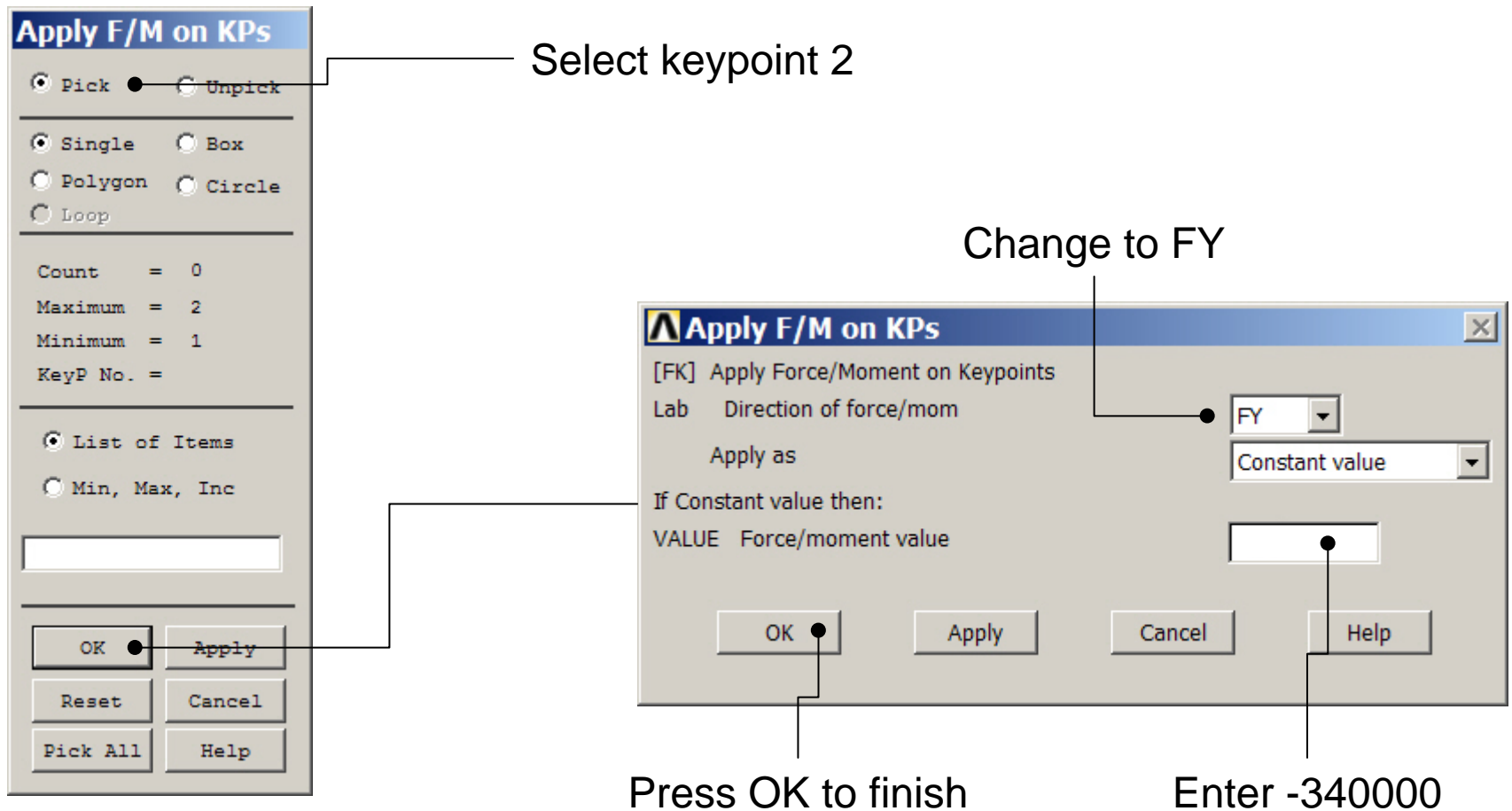
Example – Analysis Type



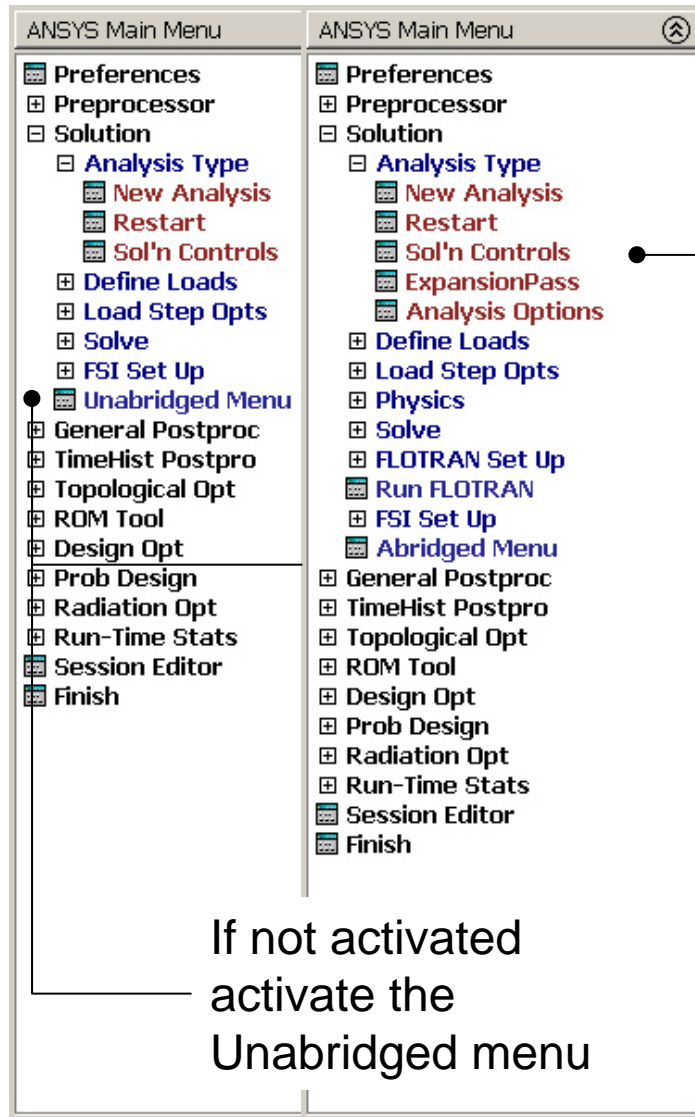
Example0511

Example – Define Loads

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



Static solution – Analysis Options



Select Sol'n Controls

If not activated
activate the
Unabridged menu

Example0511

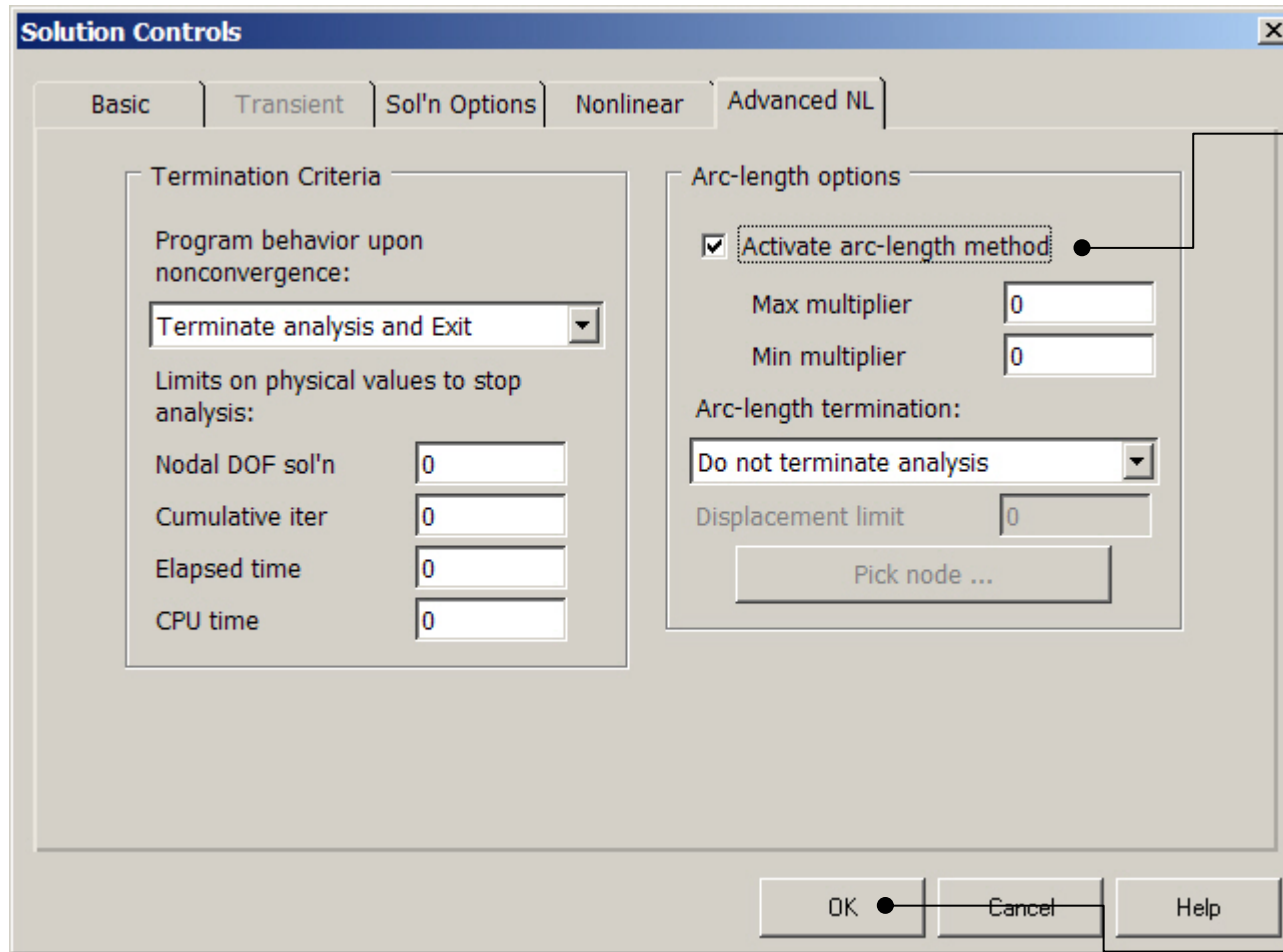
Example – Solution Controls

The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Static' selected in the dropdown menu. The 'Time Control' section has 'Time at end of loadstep' set to 0, 'Automatic time stepping' set to 'Prog Chosen', and 'Number of substeps' selected with a value of 0. The 'Write Items to Results File' section has 'All solution items' selected, and the 'Frequency' dropdown is set to 'Write last substep only'. The 'where N =' field is set to 1. Annotations with arrows point to these specific settings:

- Change to Large Displacement Static (points to the Analysis Options dropdown)
- Select All solution items (points to the 'All solution items' radio button)
- Select Write every Nth substeps (points to the Frequency dropdown)
- Enter 30 (points to the 'where N =' field)

Buttons at the bottom: OK, Cancel, Help.

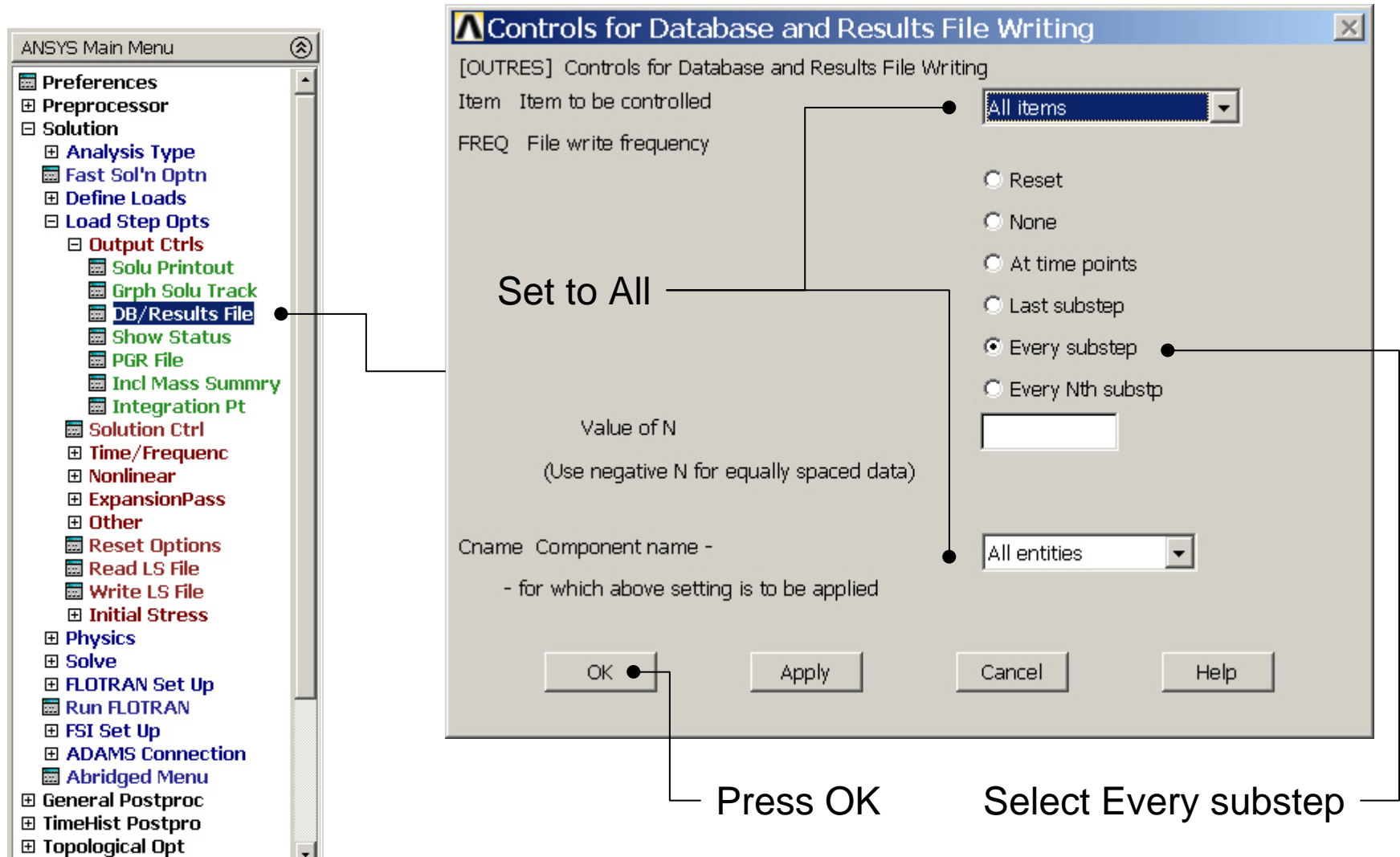
Example – Solution Controls



Activate the arc-length method

Press OK

Example – Output Ctrls

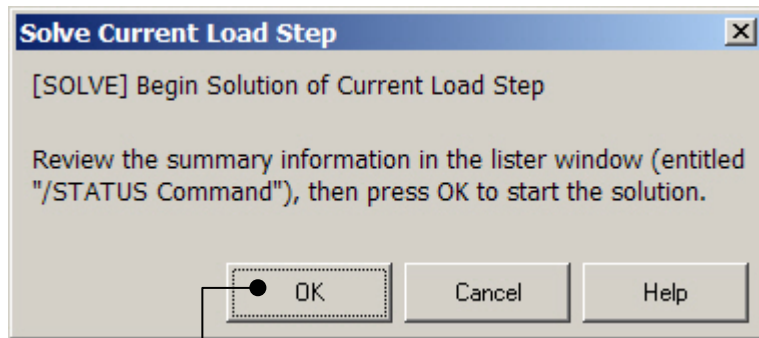


Example0511

11

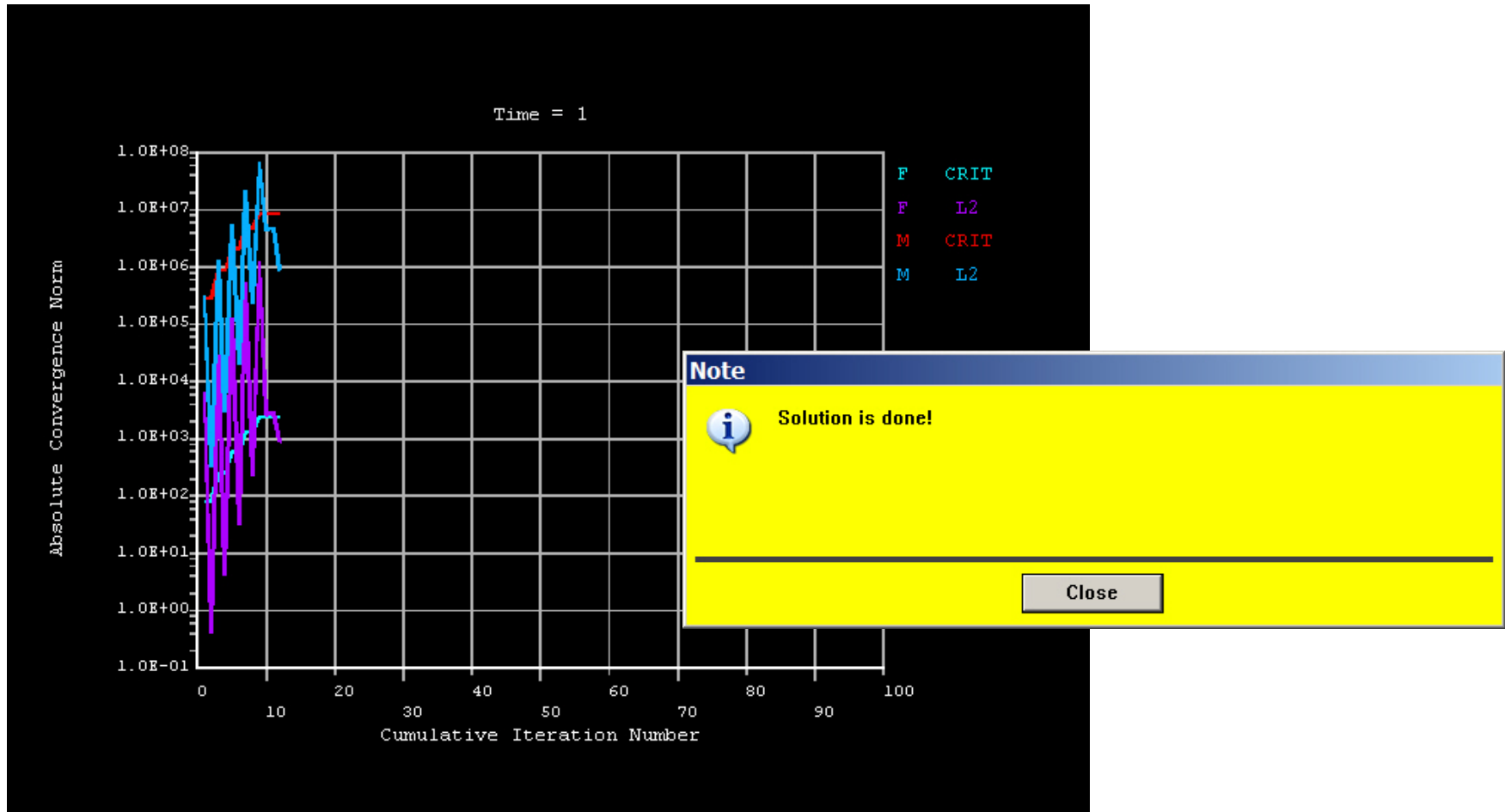
Example - Solve

Solution > Solve > Current LS

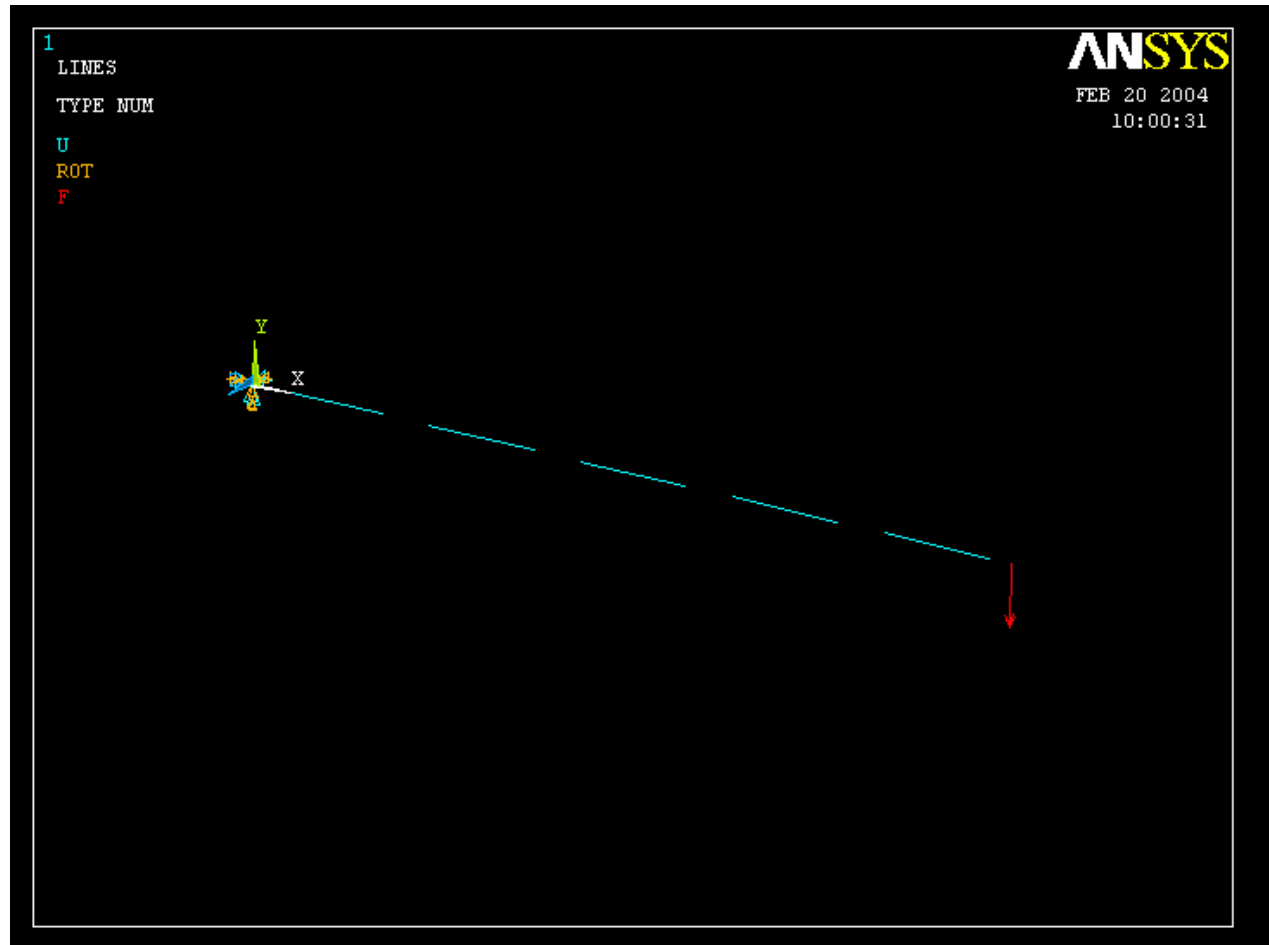
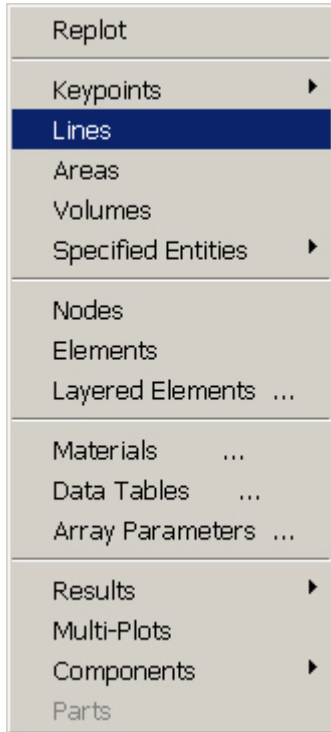


Press OK

Example - Convergence



Example – Plot - Lines



Example – TimeHistory Postpro

The image shows the ANSYS Main Menu on the left and the Time History Variables dialog box on the right. The Main Menu includes options like Preferences, Preprocessor, Solution, General Postproc, TimeHist Postpro, and Finish. The Time History Variables dialog box has a title bar 'Time History Variables - file.rst' and a menu 'File Help'. It contains a 'Variable List' table with columns: Name, Element, Node, Result Item, Minimum, Maximum, X-Axis, and a selection icon. The table has one row: 'TIME', 'Time', '1', '1', and 'X-Axis' is selected. Below the table is a 'Calculator' section with a display area and a keypad with various mathematical functions and operators. A line points from the text 'Close this dialog box' to the close button (X) in the top right corner of the dialog box.

ANSYS Main Menu

- Preferences
- Preprocessor
- Solution
- General Postproc
- TimeHist Postpro**
 - Variable Viewer
 - Settings
 - Store Data
 - Define Variables
 - Read LSDYNA Data
 - List Variables
 - List Extremes
 - Graph Variables
 - Math Operations
 - Table Operations
 - Smooth Data
 - Generate Spectrm
 - Reset Postproc
- Topological Opt
- ROM Tool
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
- Finish

Time History Variables - file.rst

File Help

Variable List

Name	Element	Node	Result Item	Minimum	Maximum	X-Axis	
TIME			Time	1	1	X-Axis	

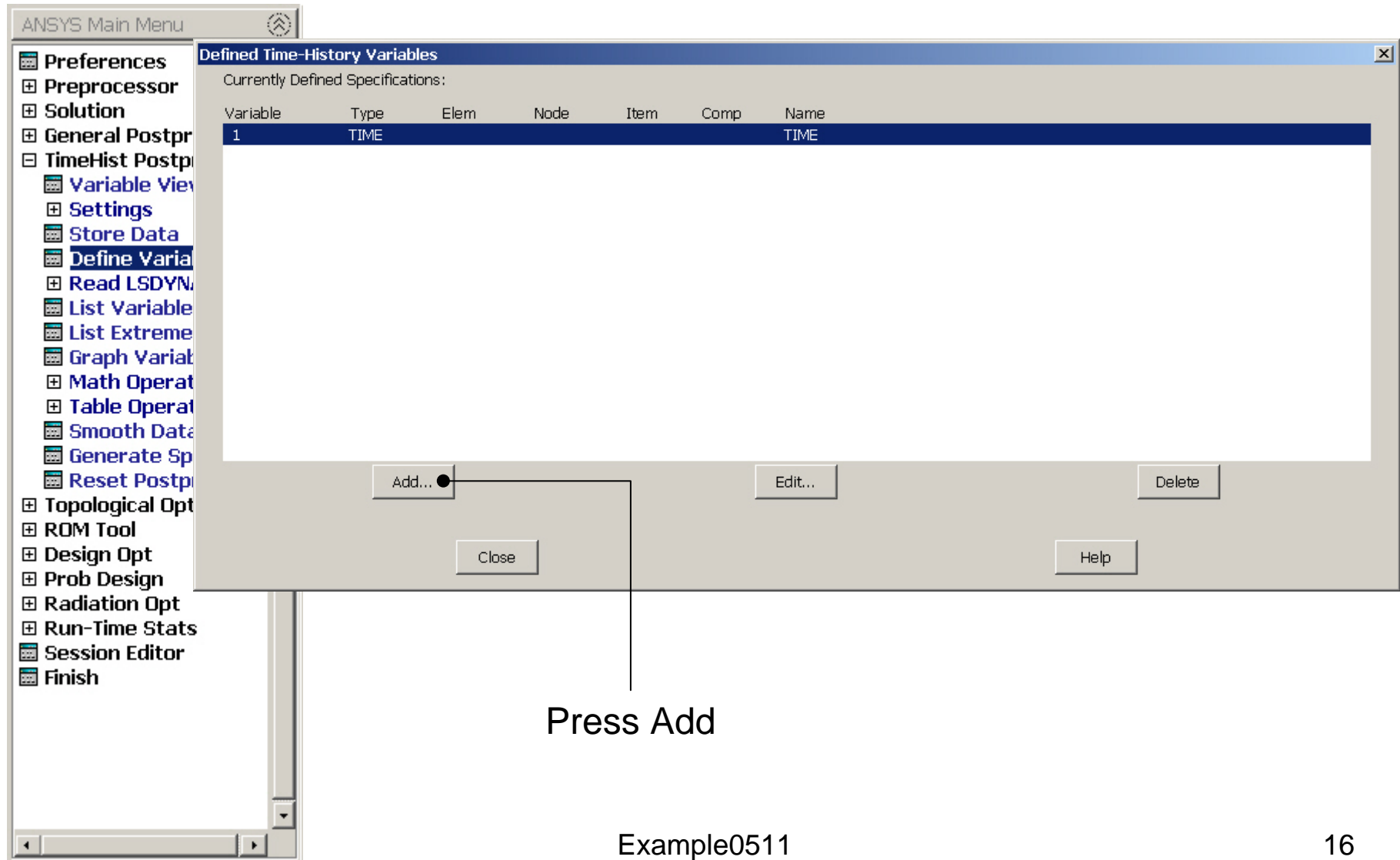
Calculator

Close this dialog box

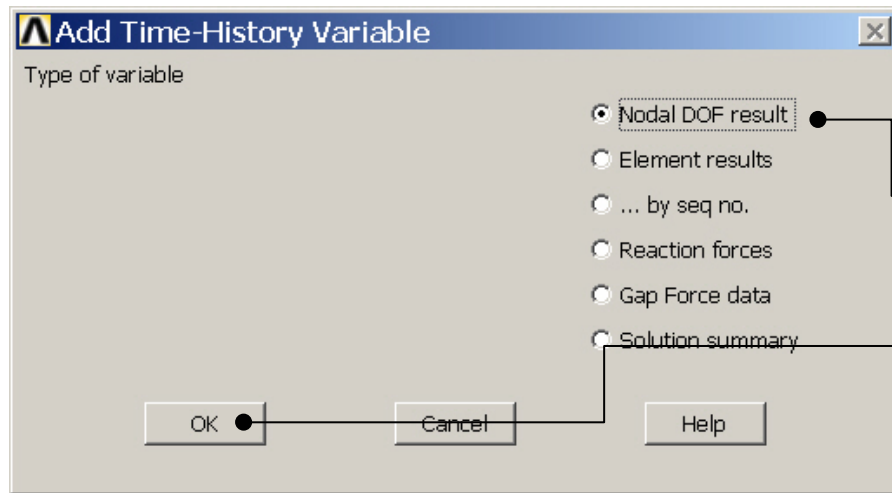
Example0511

15

Example – Define Variables



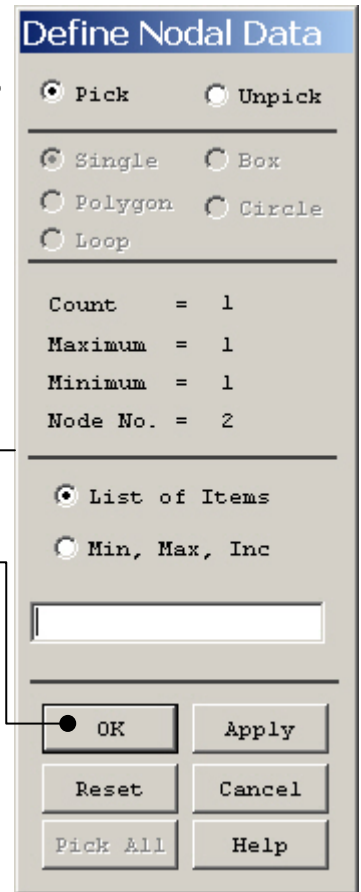
Example – Add Time-History Var.



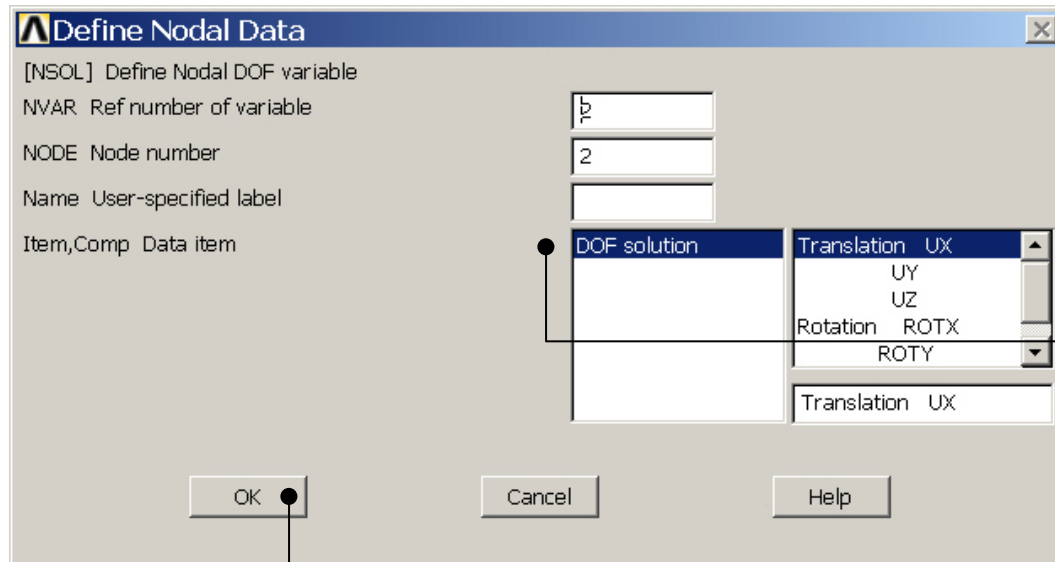
Pick the right node

Select Nodal
DOF result

Press OK

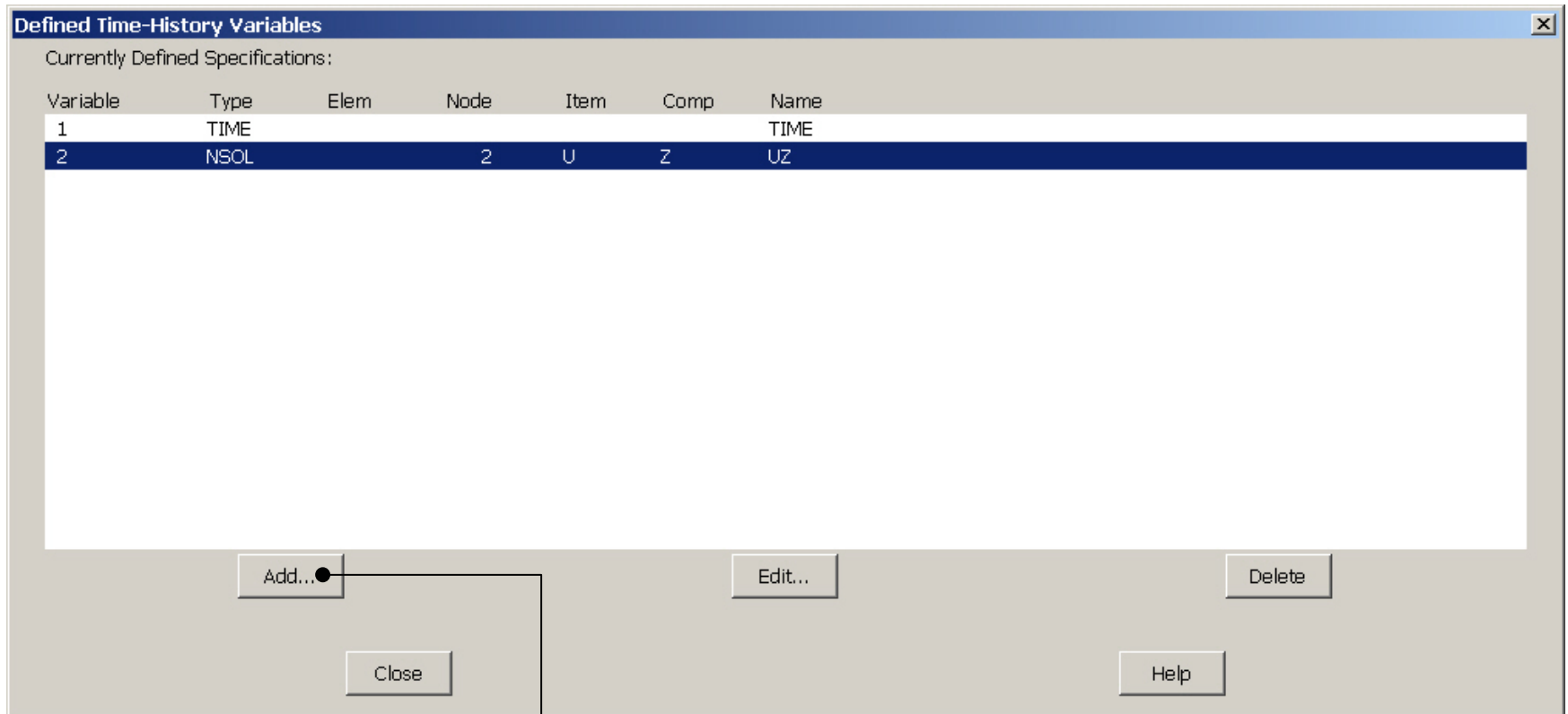


Press OK

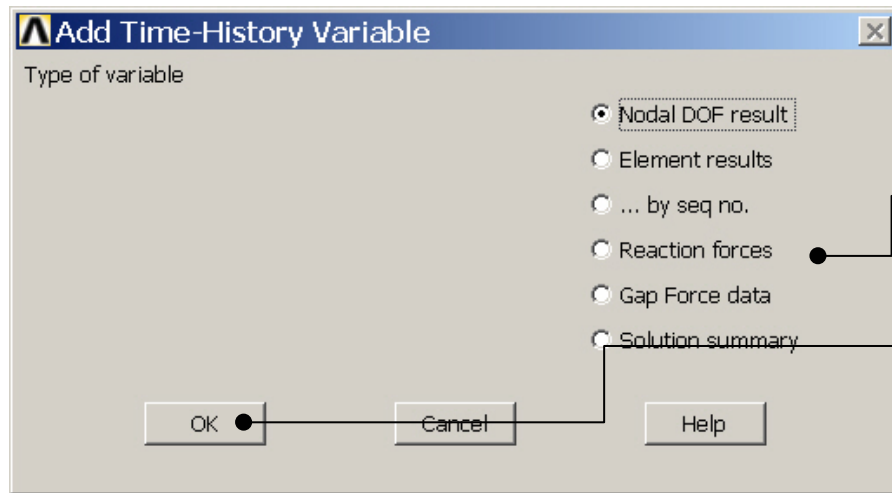


Select DOF solution
and Translation UZ

Example – Add Time-History Var.



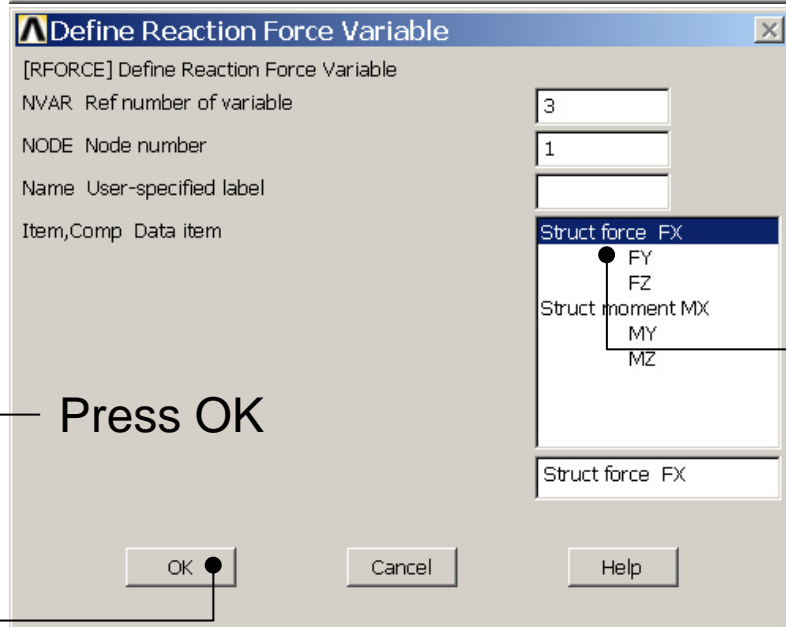
Example – Add Time-History Var.



Pick the left node

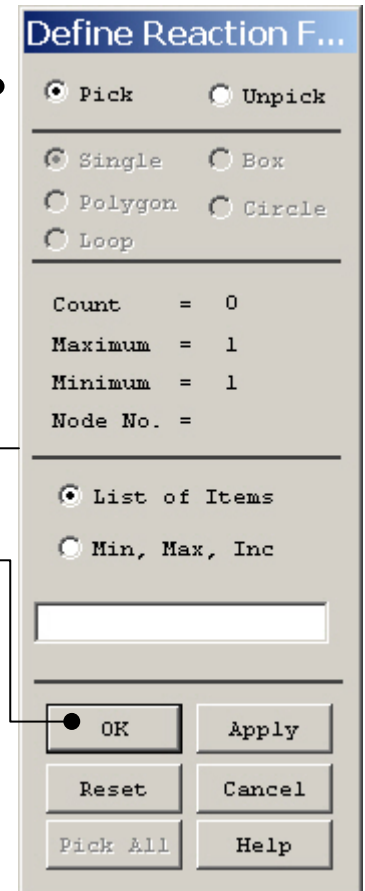
Select Reaction forces

Press OK



Press OK

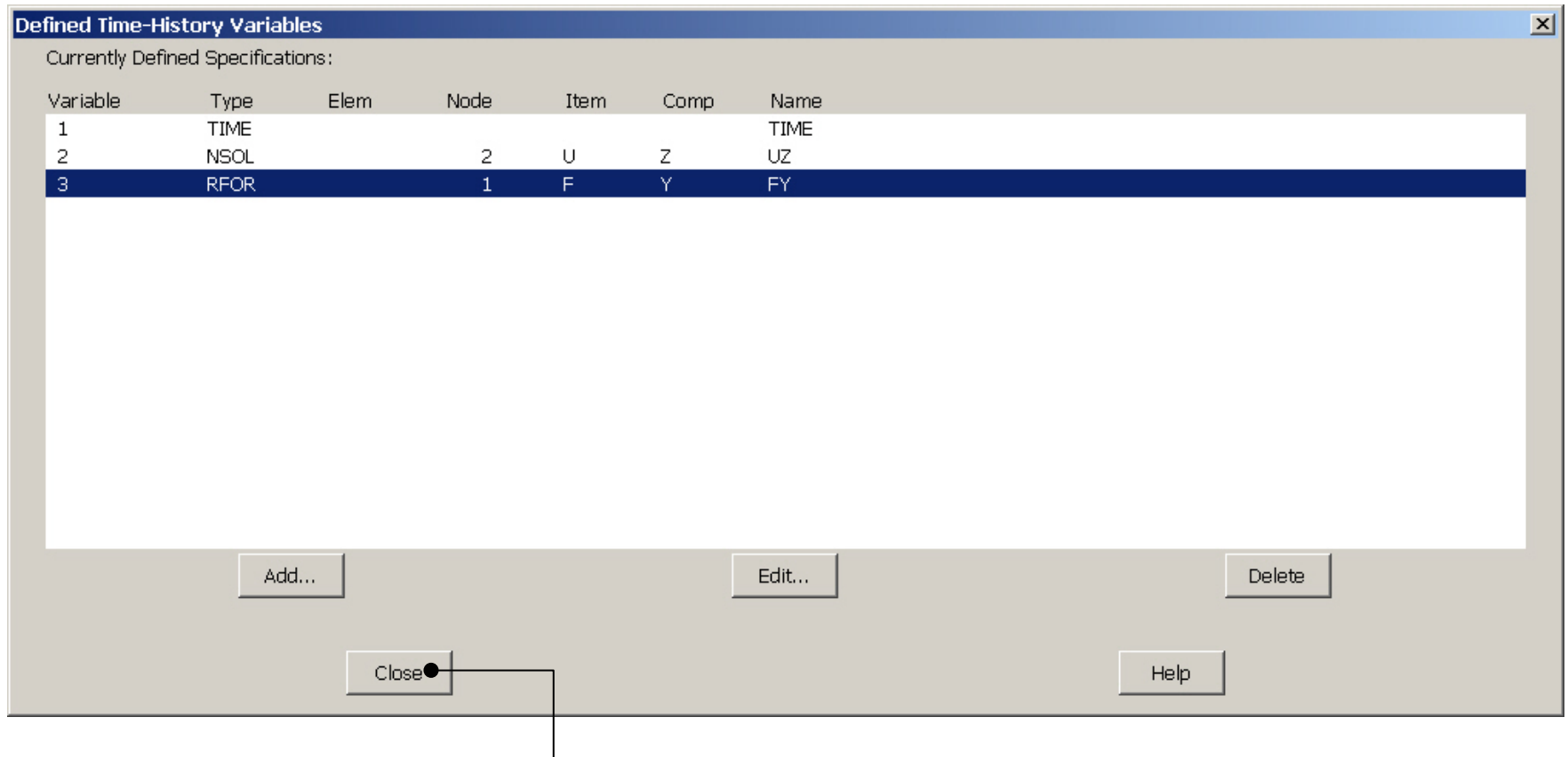
Select Struct force FY



Press OK

Example0511

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)

Example0511

21

Example – Style - Graph

The image shows the ANSYS software interface with the 'Style' menu open. The 'Style' menu is highlighted, and the 'Graphs' option is selected. The 'Graphs' submenu is also open, showing options like 'Viewing Control', 'Modify Curve ...', 'Modify Grid ...', 'Modify Axes ...', and 'Select Anno/Gr'. The 'Modify Axes ...' option is highlighted. The 'Axes Modifications for Graph Plots' dialog box is open, showing various settings for the graph axes. The 'X-axis label' is set to '/AXLAB', the 'Y-axis label' is set to '/AXLAB', and the 'Thickness of axes' is set to 'Double'. The 'Number of Y-axes' is set to 'Single Y-axis'. The 'X-axis range' is set to 'Auto calculated'. The 'Y-axis range' is set to 'Auto calculated'. The 'Specified X range' is set to 'XMIN,XMAX'. The 'Specified Y range' is set to 'YMIN,YMAX'. The 'Y-axis number' is set to '1'. The 'Y ranges for' is set to 'Individual calcs'. The 'Axis Controls' section shows 'LOGX' set to 'Linear', 'LOGY' set to 'Linear', 'AXDV' set to 'On', 'AXNM' set to 'On - back plane', 'AXNSC' set to '1', 'DIG1' set to '4', 'DIG2' set to '3', and 'XAXO' set to '0'. The 'OK' button is highlighted.

Enter Deformation UZ

Enter Force FY

Press OK

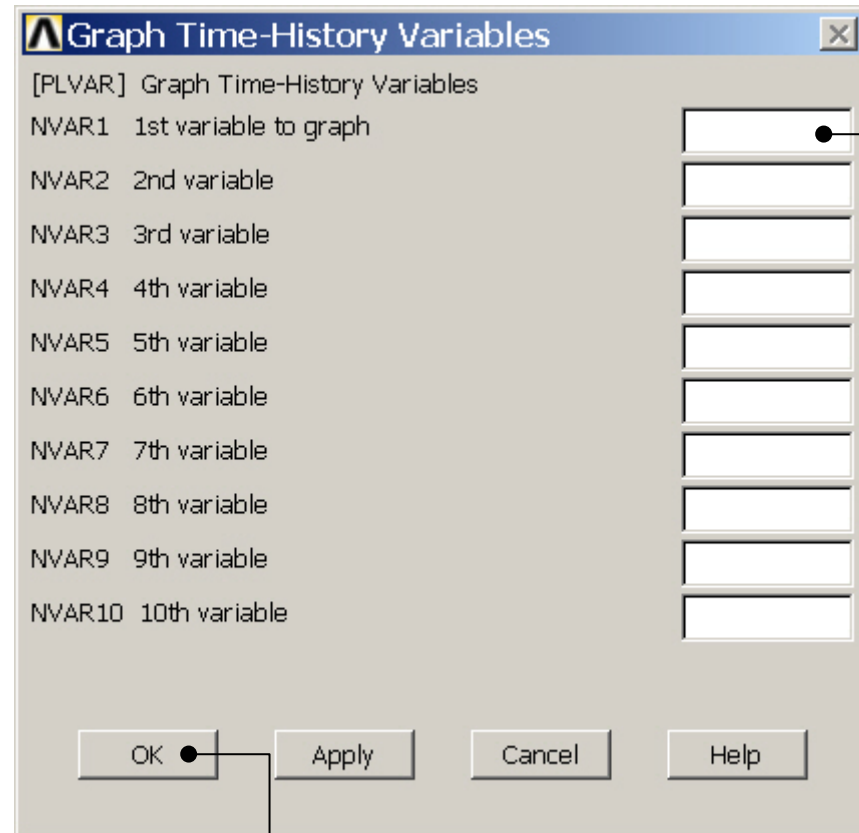
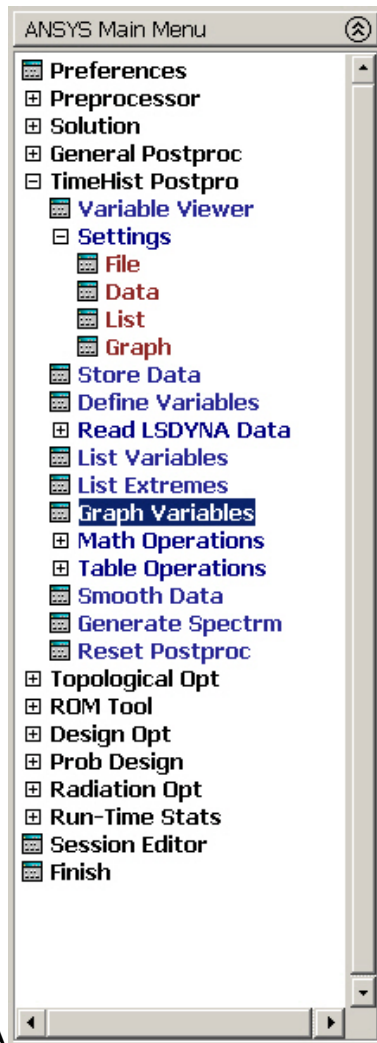
ANSYS

Computational Mechanics, AAU, Esbjerg

Example0511

22

Example – Graph Variables

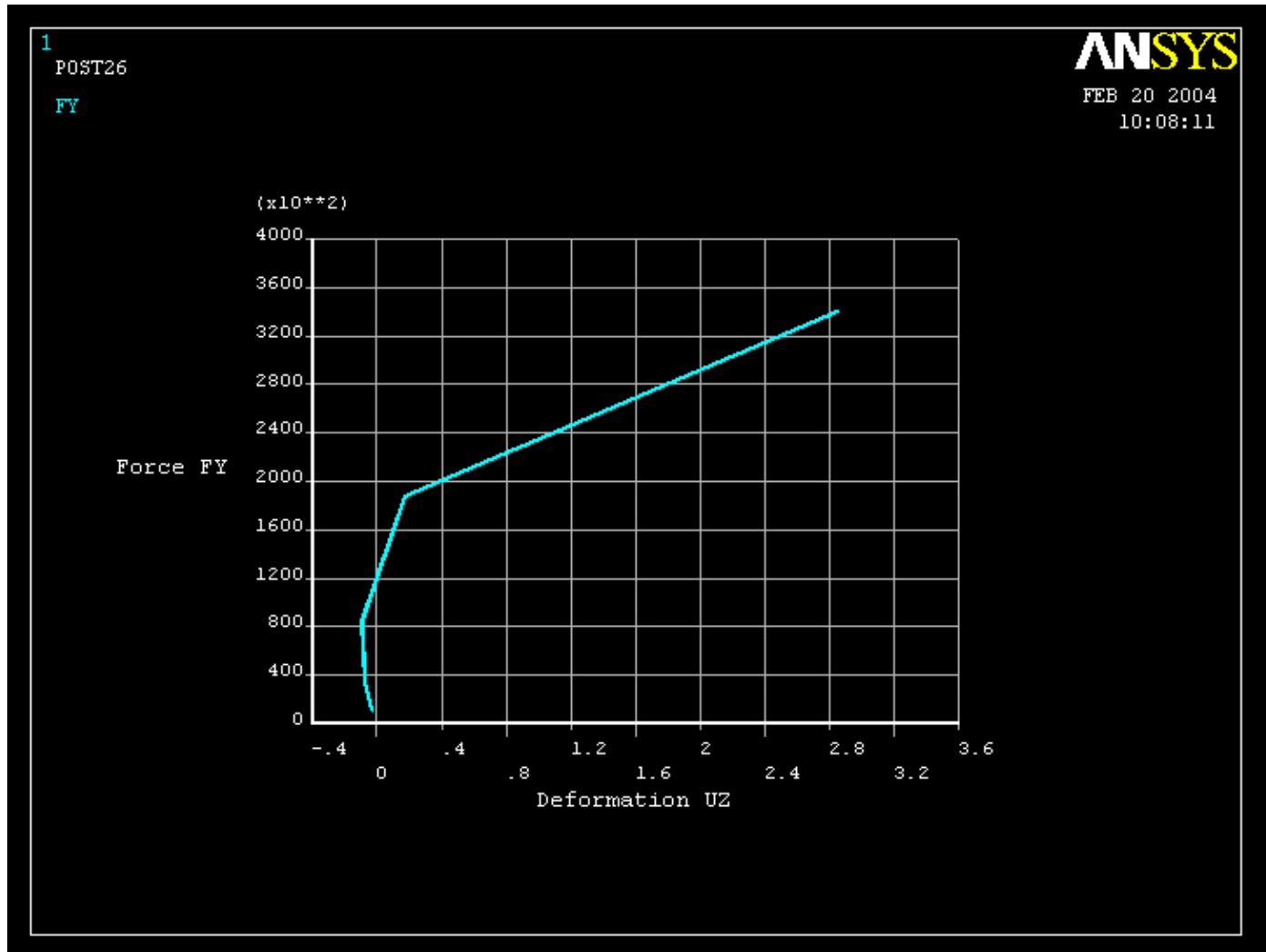


Enter 3 to plot
the reaction
force FY on the
Y-axis

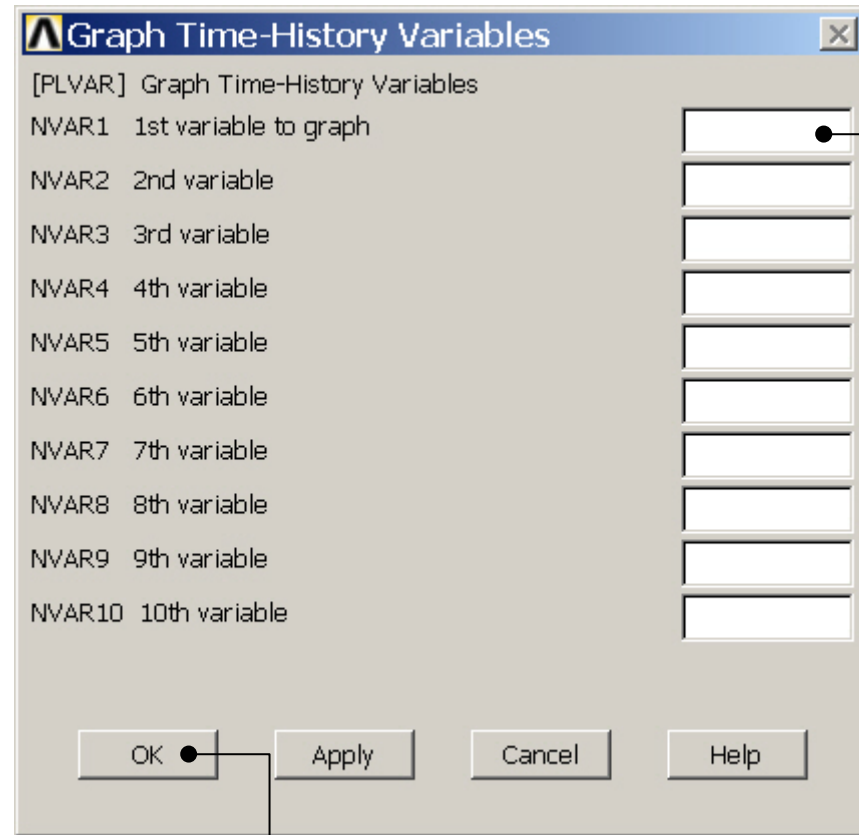
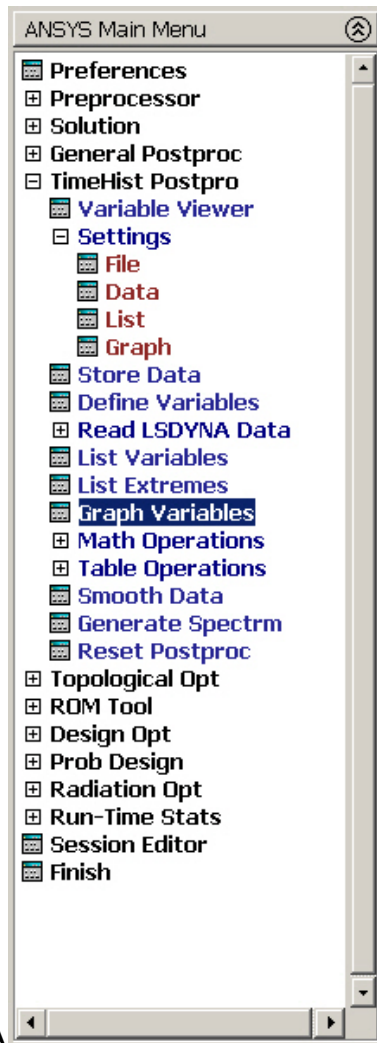
Press OK

Example0511

Example - Graph



Example – Graph Variables



Enter 3 to plot
the reaction
force FY on the
Y-axis

Example0511