

Course in ANSYS

Example0510

Example – Column beam



Objective:

Plot the P-U curve for the nonlinear behaviour

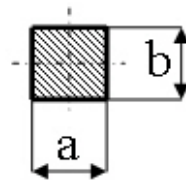
Tasks:

Obtain a static solution including prestress?

Obtain a buckling solution?

Include imperfections using Update Geometry?

Run the nonlinear analysis



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

$$\nu = 0.3$$

$$L = 1000 \text{ mm}$$

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

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

$$F = 430 \text{ N}$$

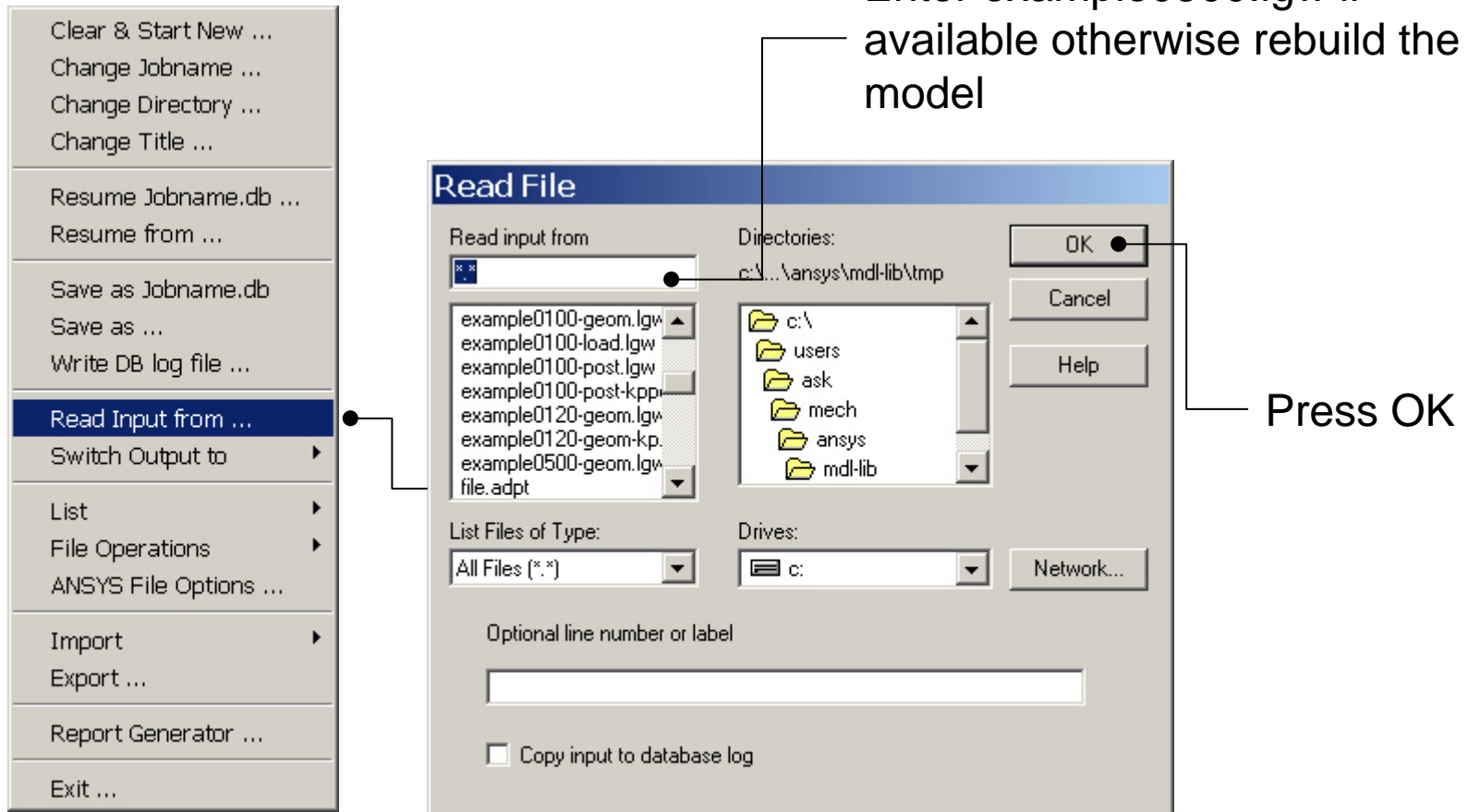
Nonlinear - Solution Phases

- 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

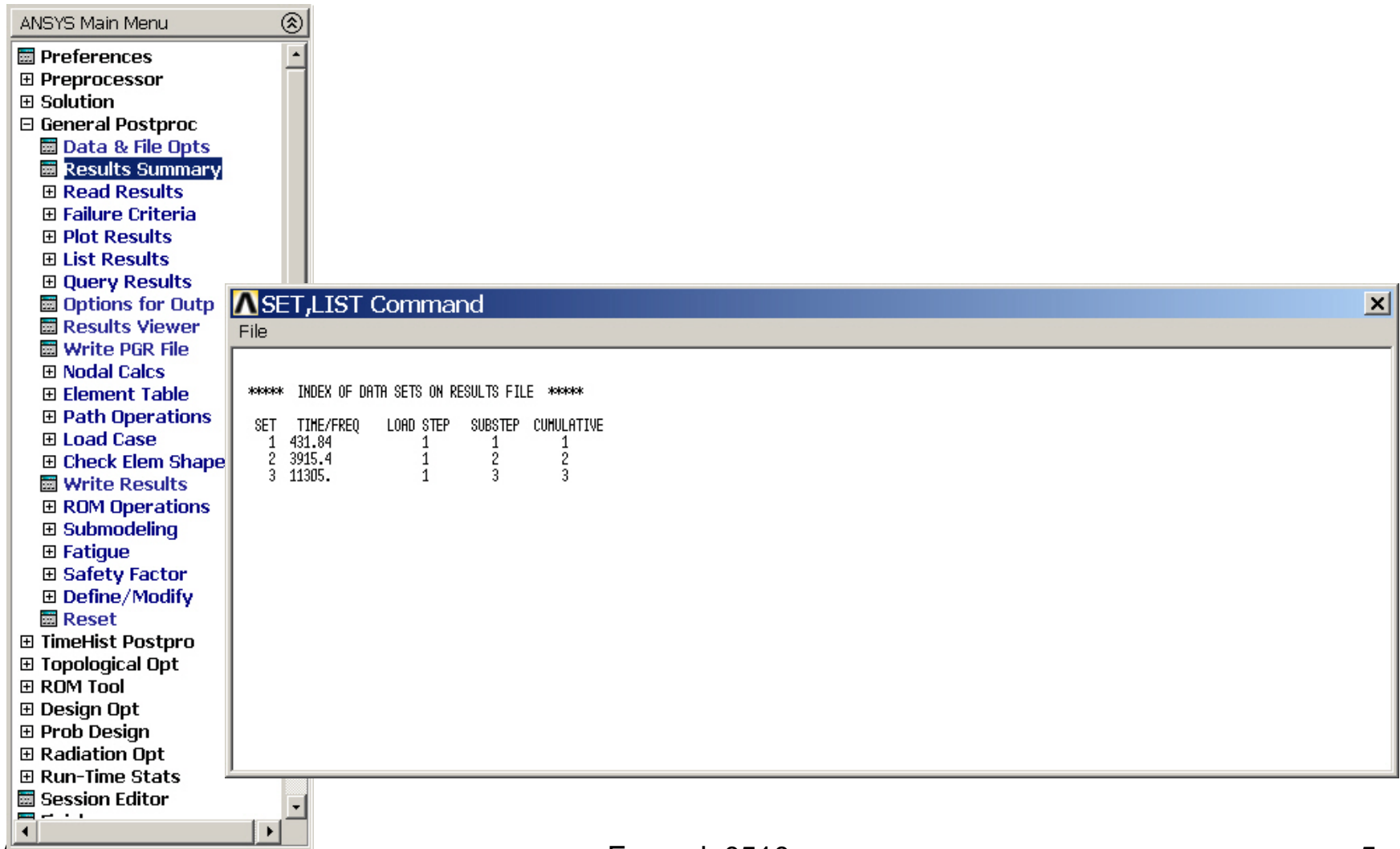
We start
here



Example – Read Input from..

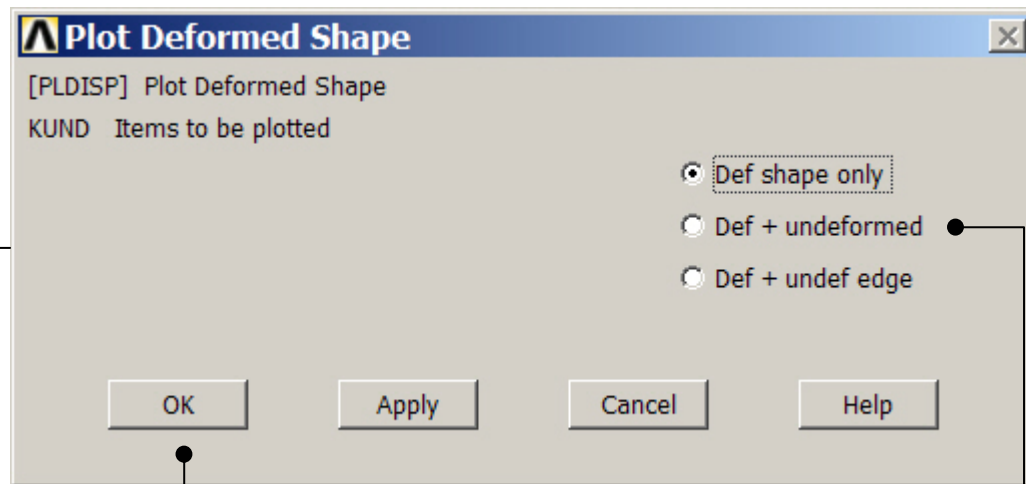
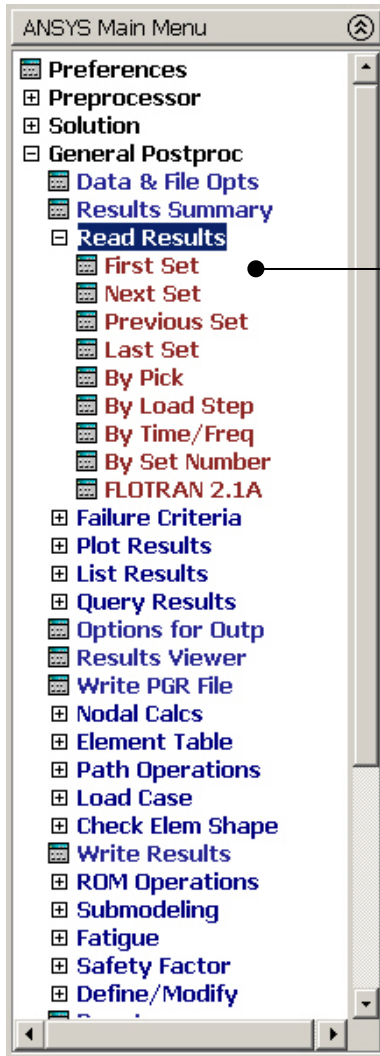


Example – Results Summary



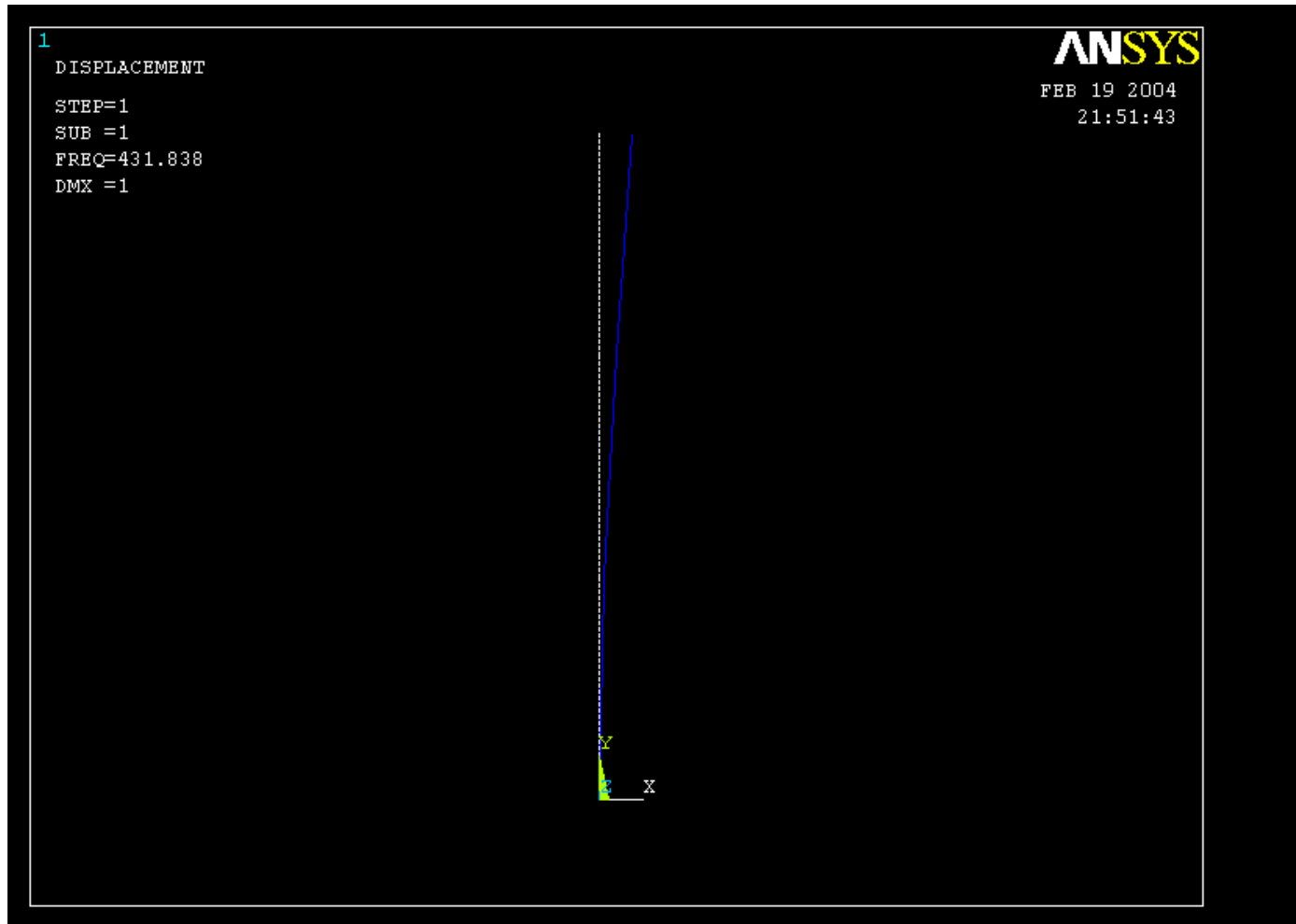
Example – Read Results

General Postproc > Plot Results > Deformed Shape

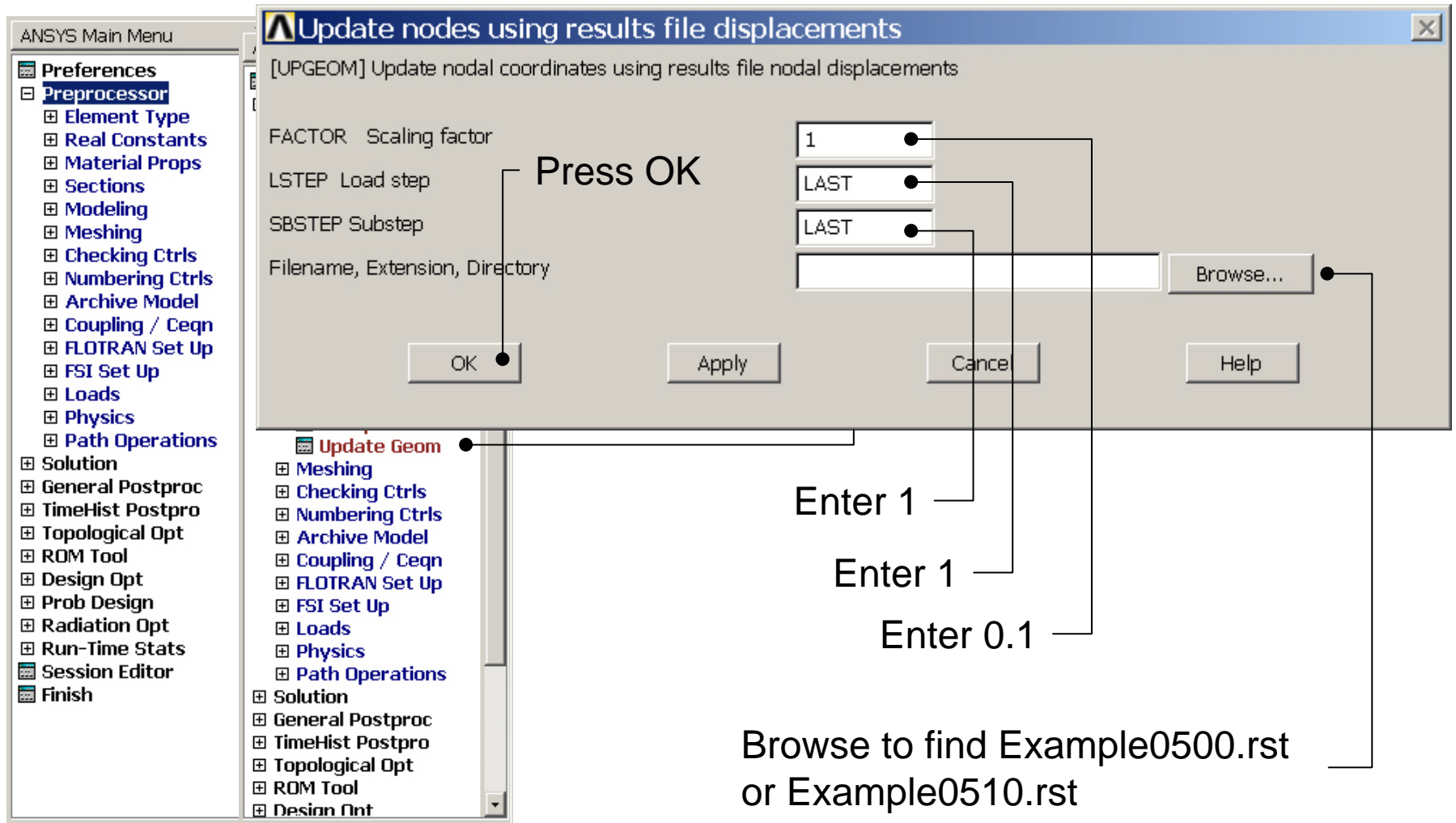


Select "Def+undeformed"
and Press OK

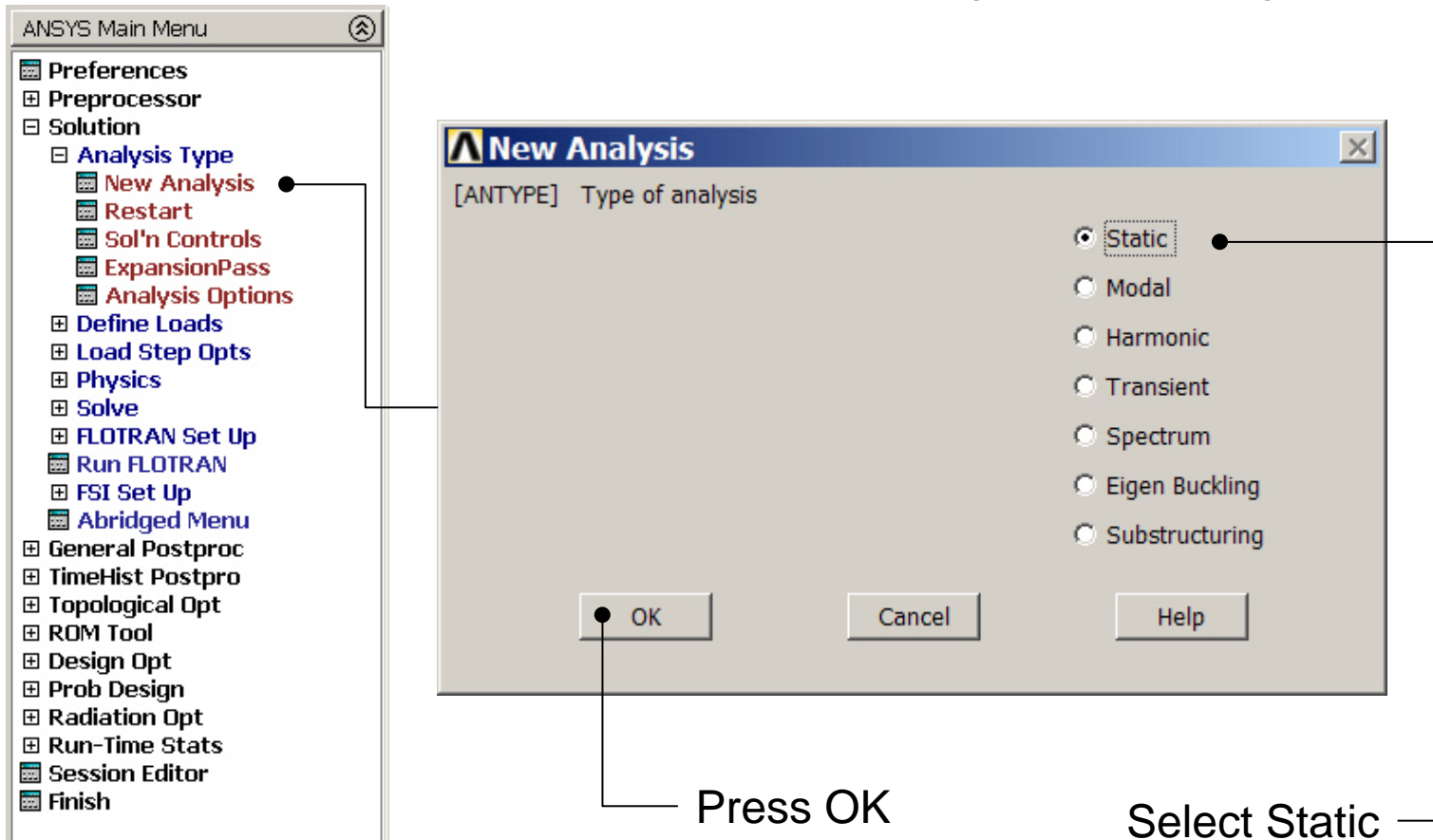
Example – Buckling mode



Example - Update Geom



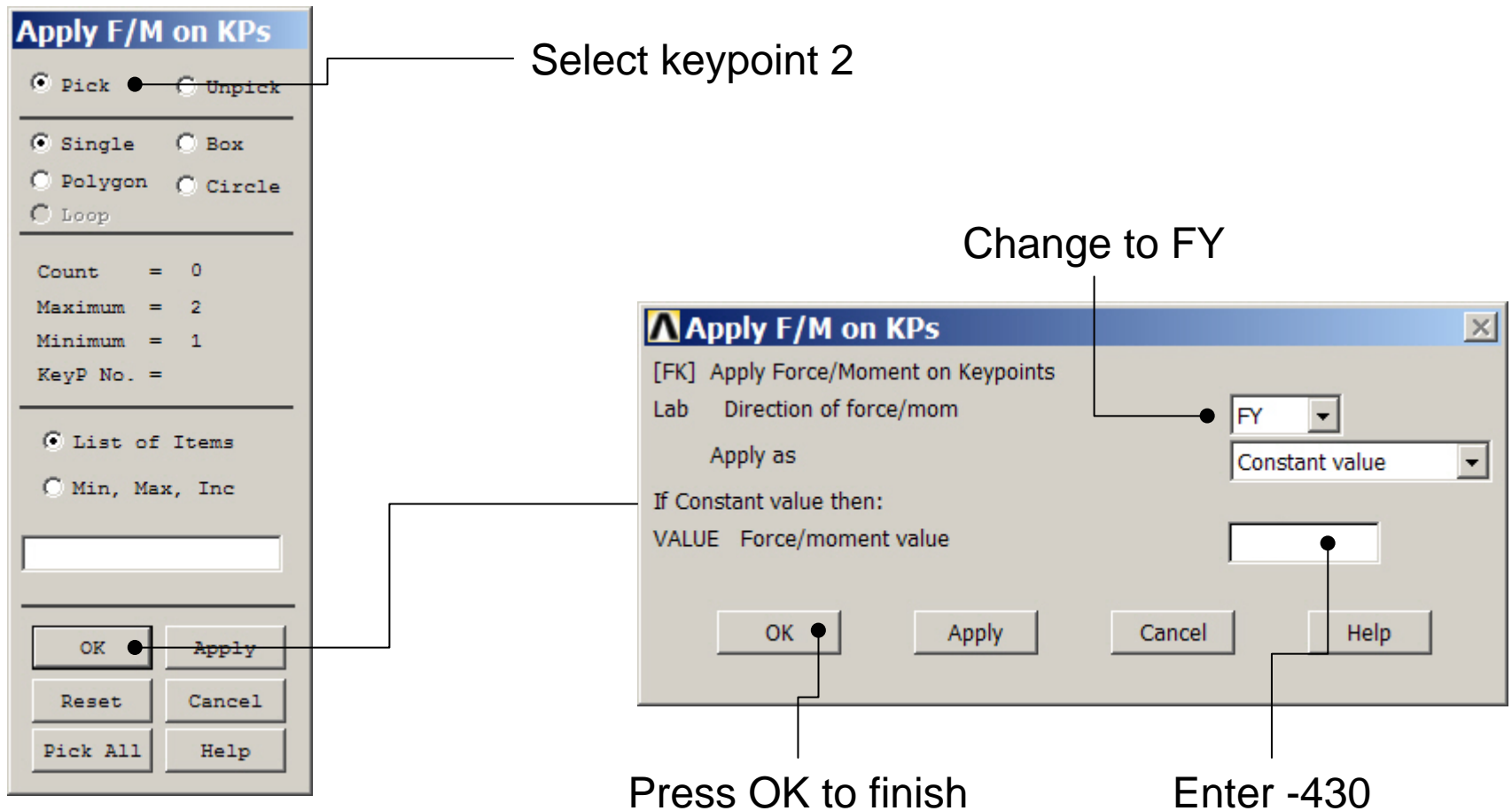
Example – Analysis Type



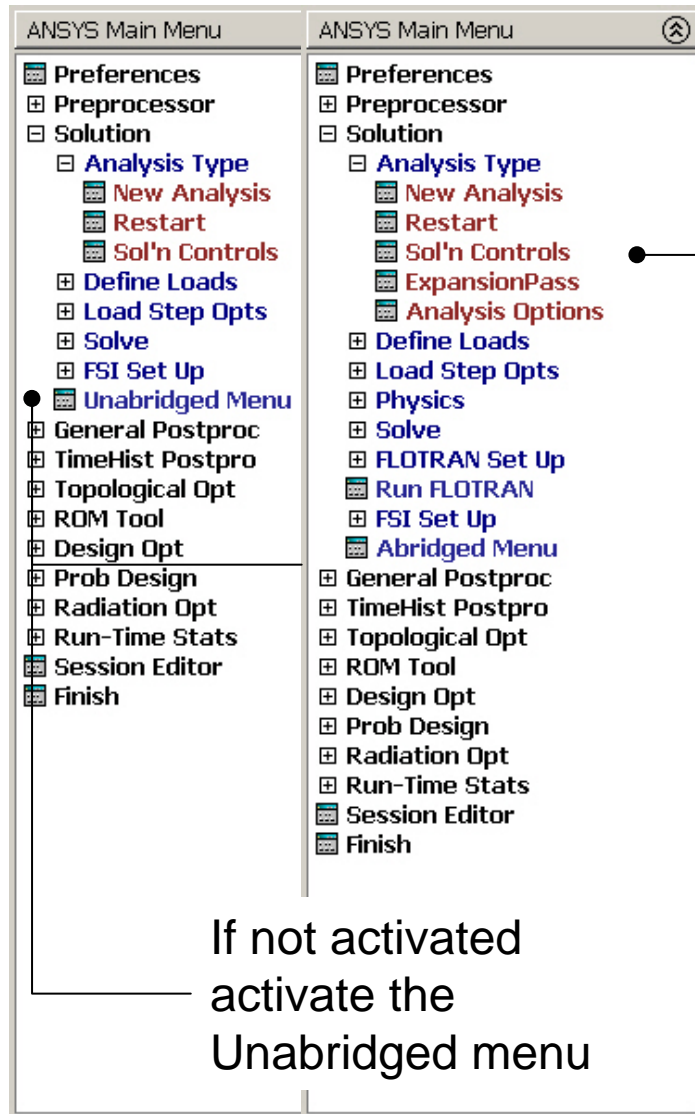
Example0510

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

Example0510

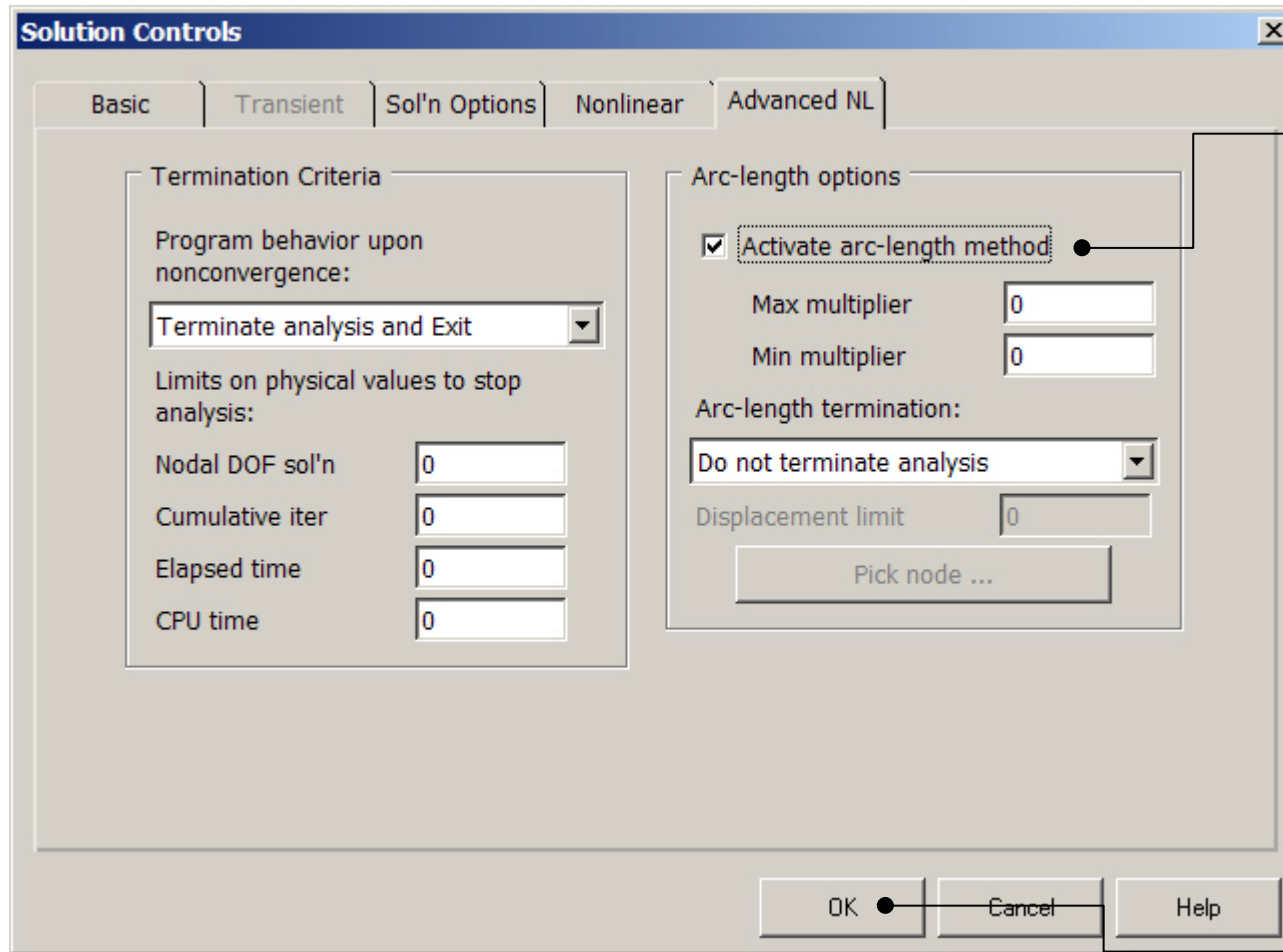
Example – Solution Controls

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 the value 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.

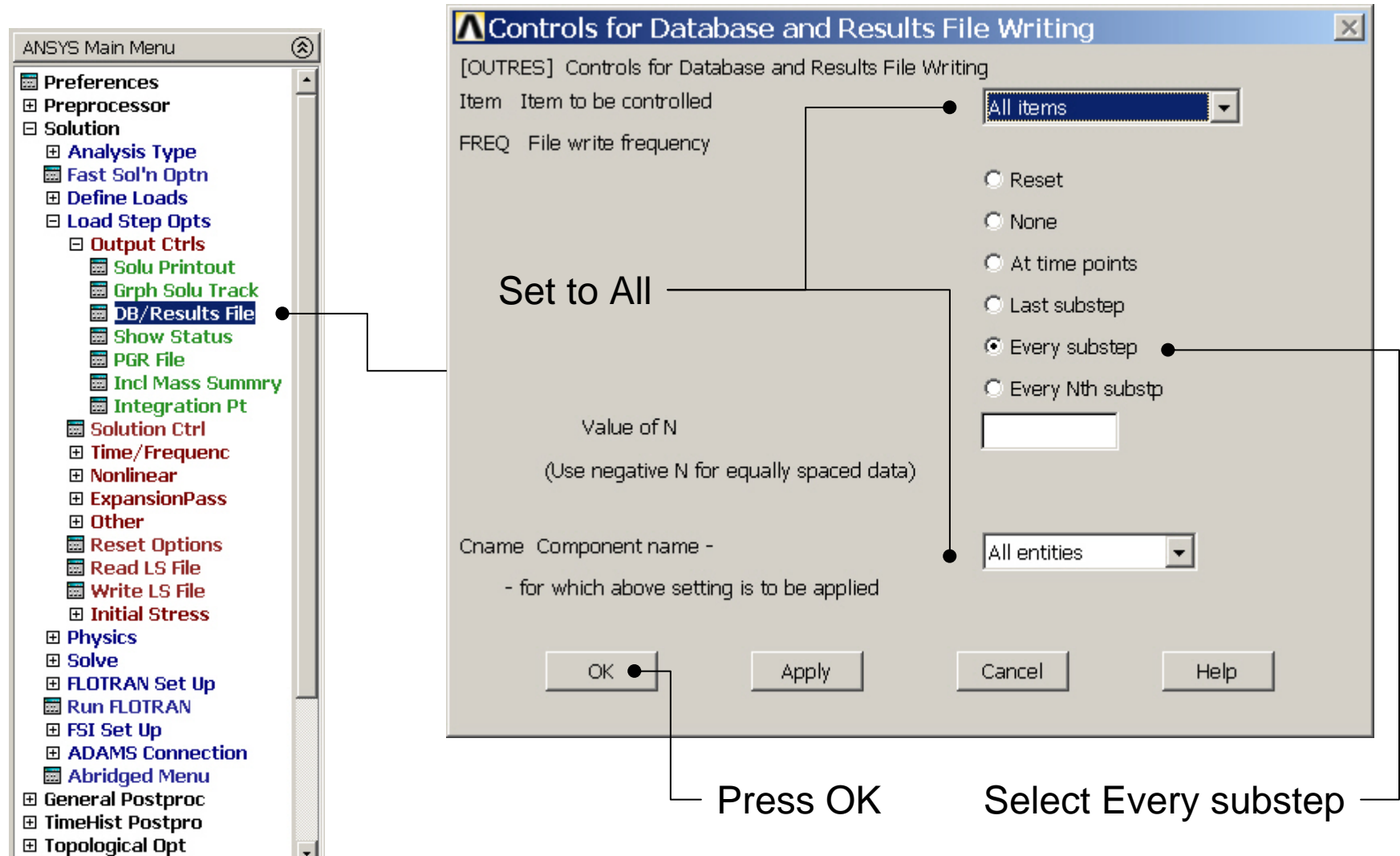
Example – Solution Controls



Activate the arc-length method

Press OK

Example – Output Ctrls

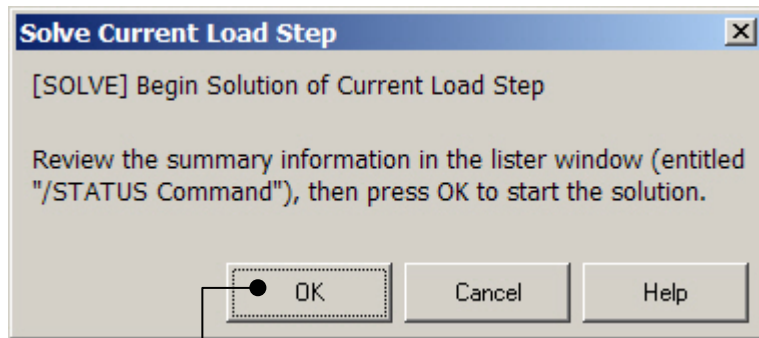


Example0510

14

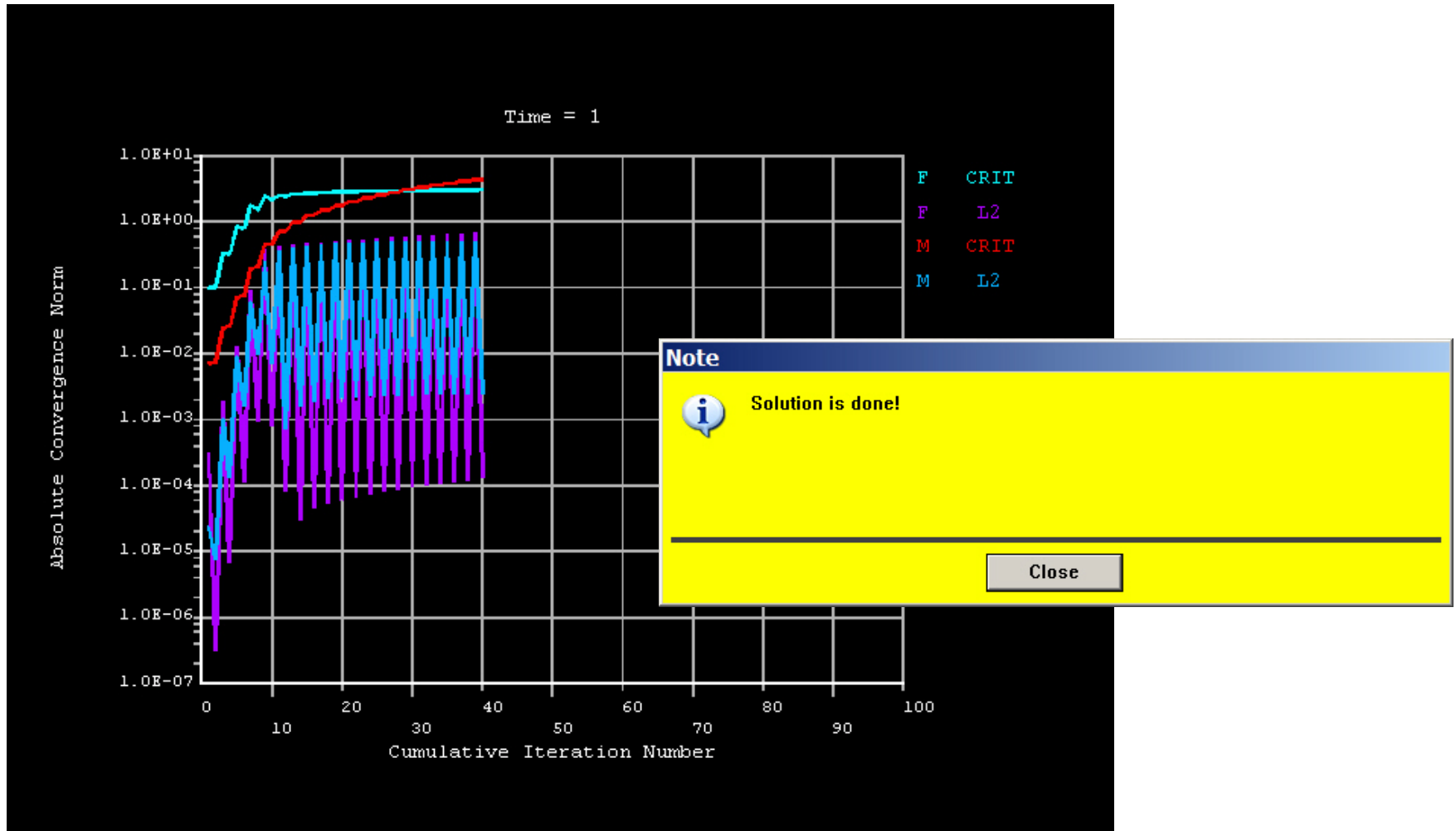
Example - Solve

Solution > Solve > Current LS

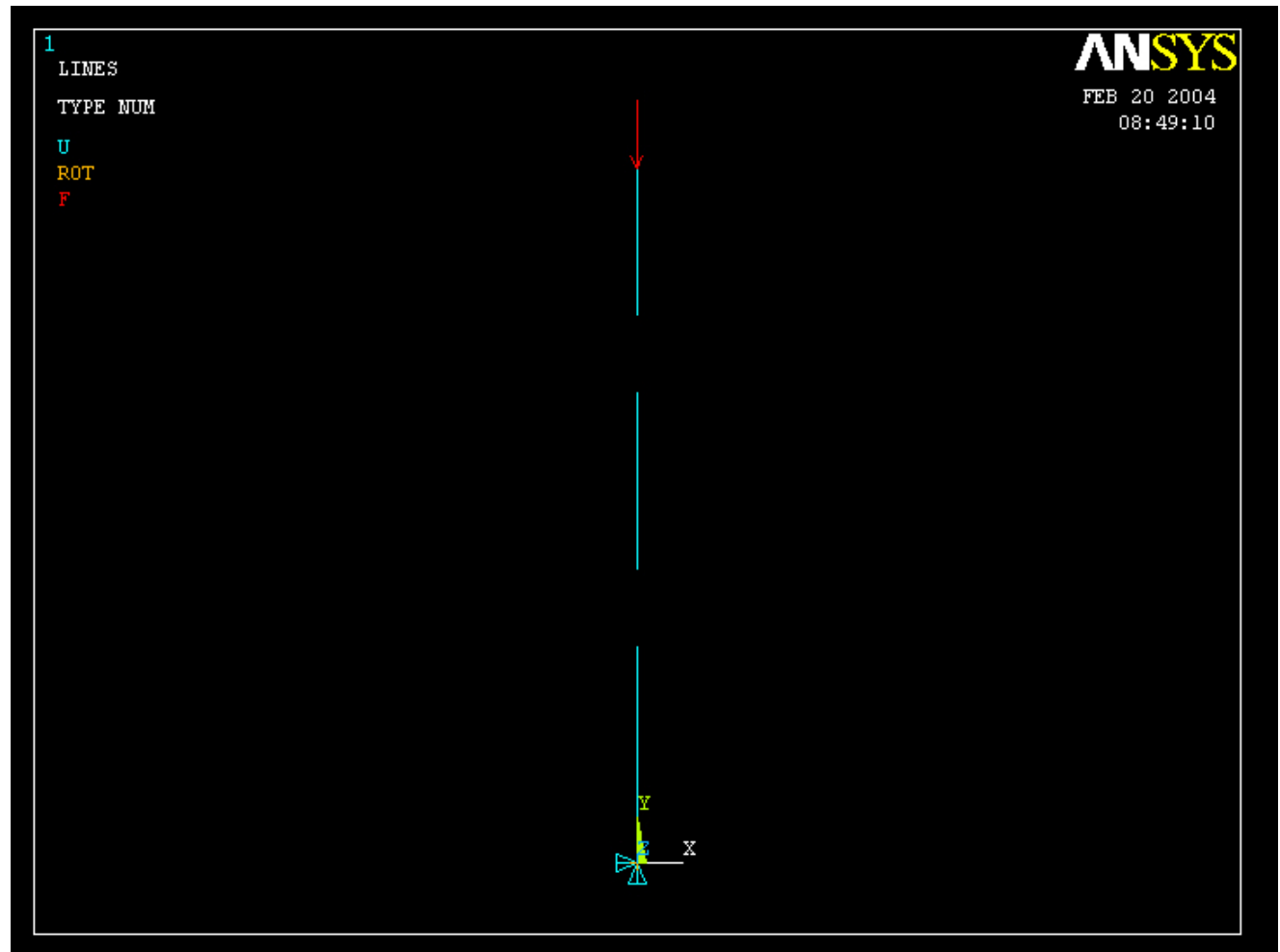
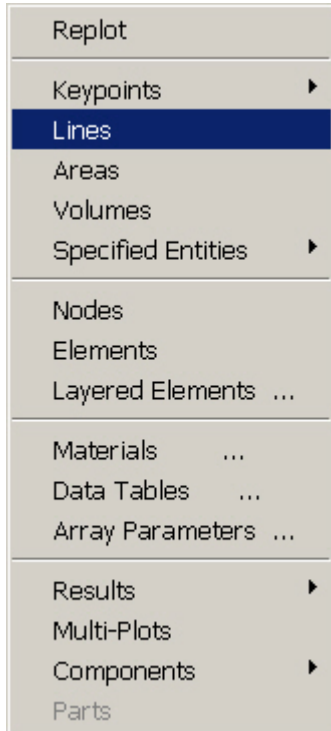


Press OK

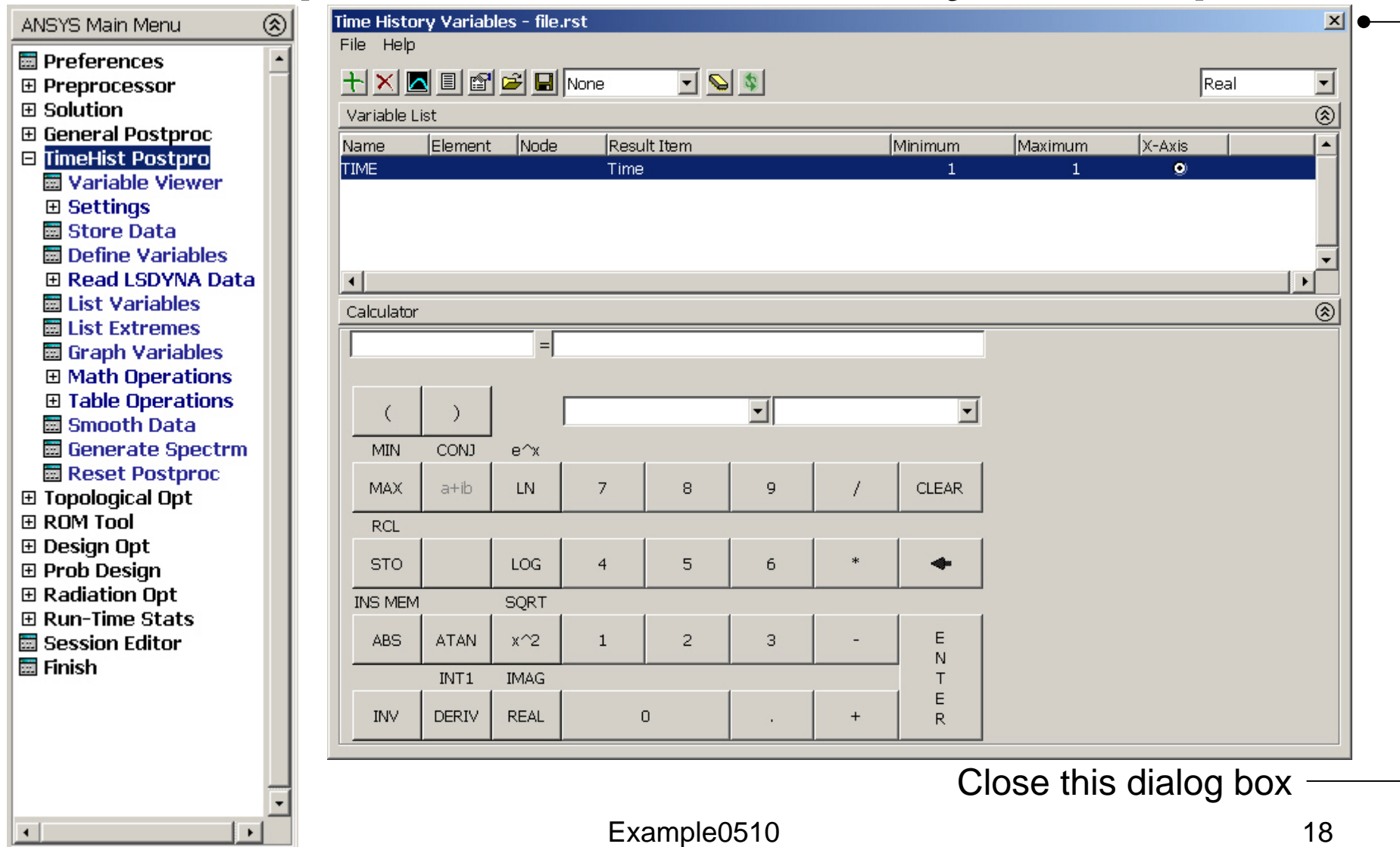
Example - Convergence



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 dialog box is titled "Time History Variables - file.rst" and contains a "Variable List" table and a "Calculator" section.

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	○

Calculator

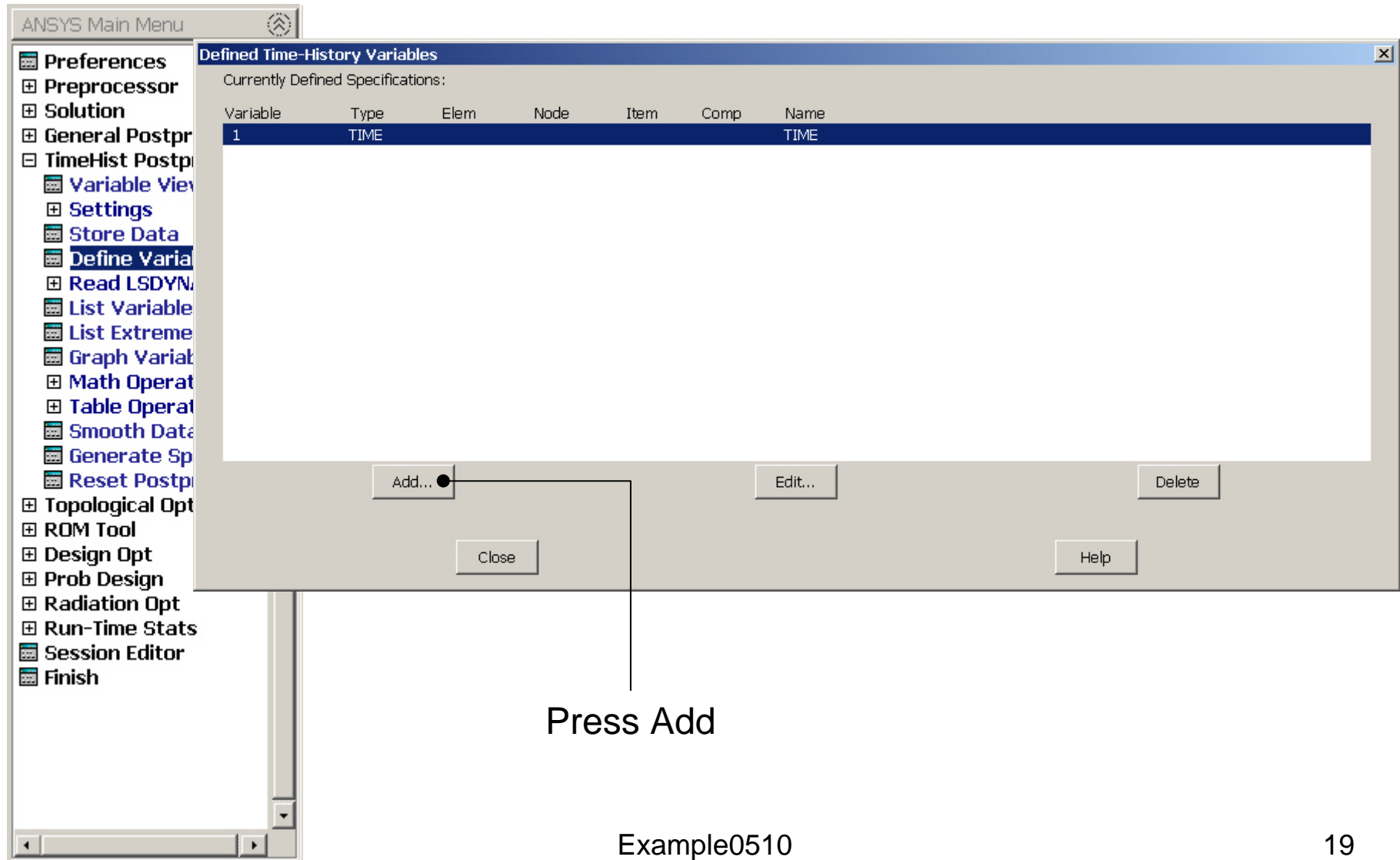
Calculator interface with buttons for MIN, MAX, STO, ABS, INV, CONJ, ATAN, DERIV, e^x, LN, INT1, SQRT, IMAG, REAL, and a numeric keypad.

Close this dialog box

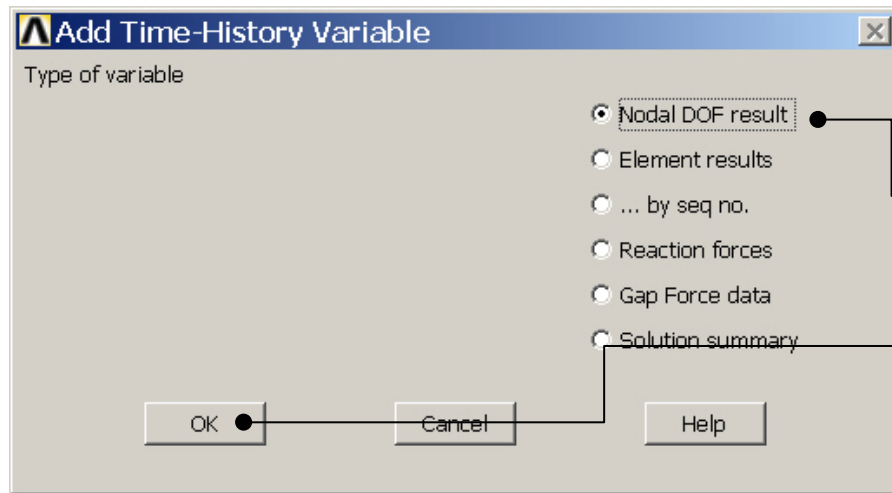
Example0510

18

Example – Define Variables



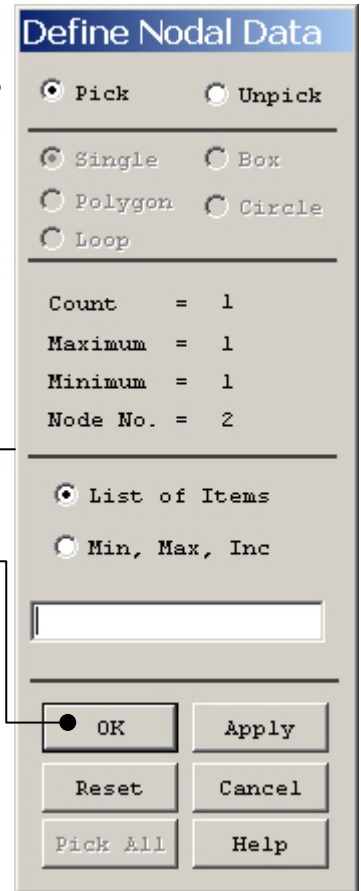
Example – Add Time-History Var.



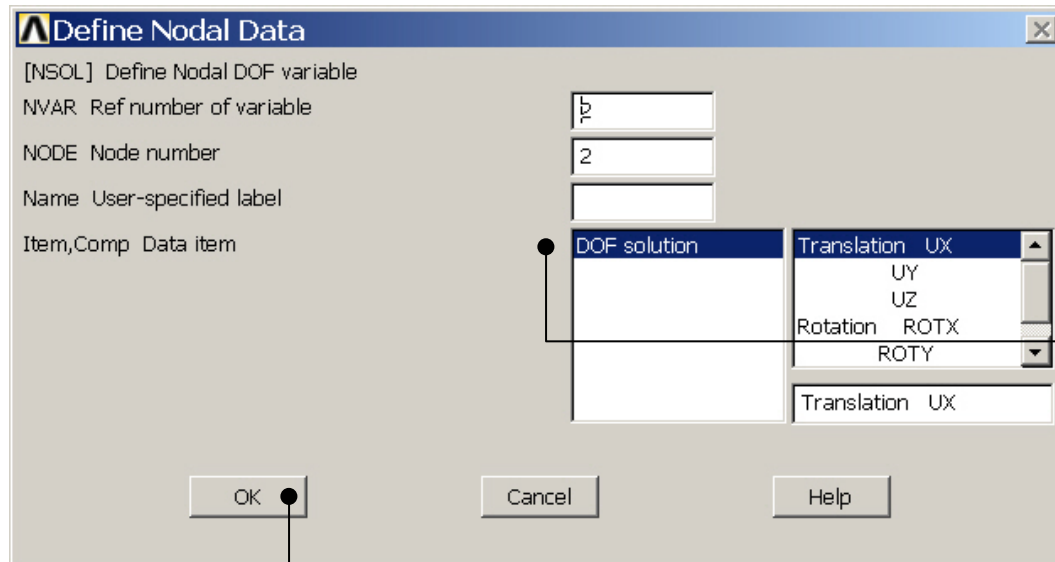
Pick the top node

Select Nodal
DOF result

Press OK

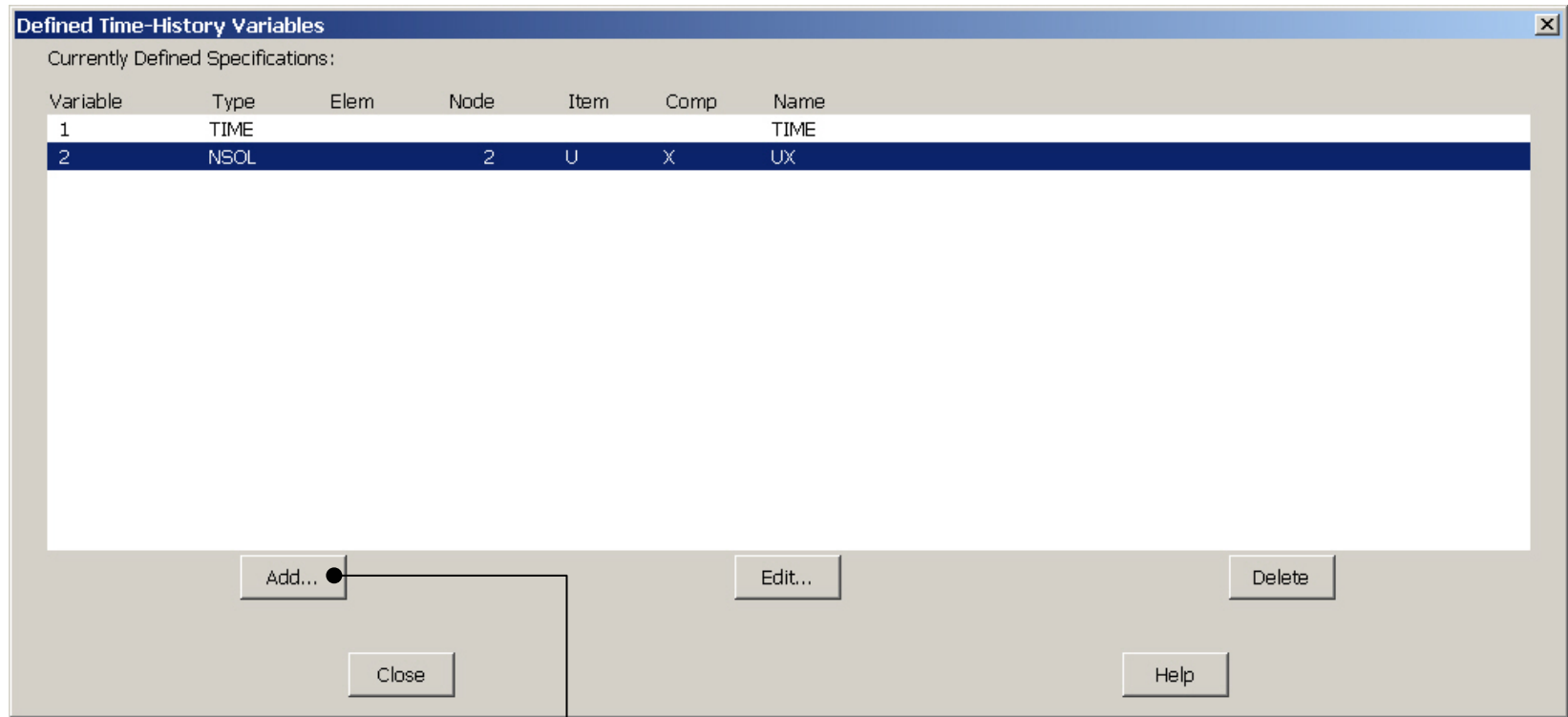


Press OK

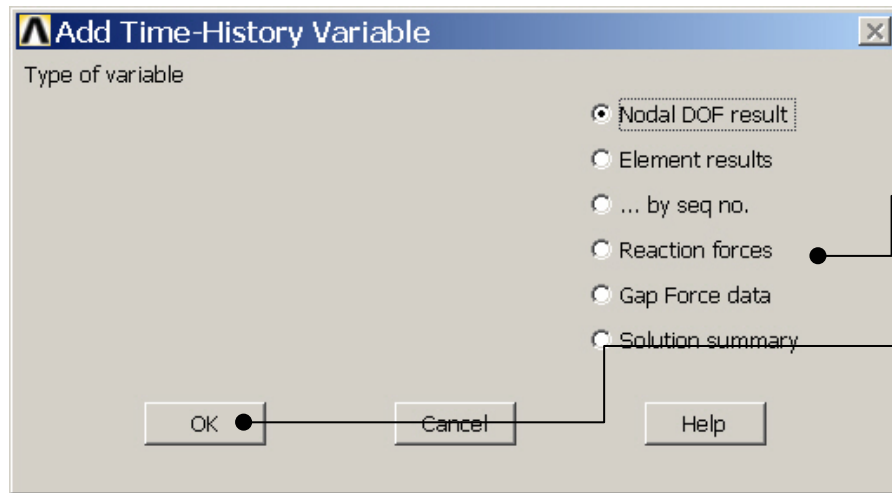


Select DOF solution
and Translation UX

Example – Add Time-History Var.



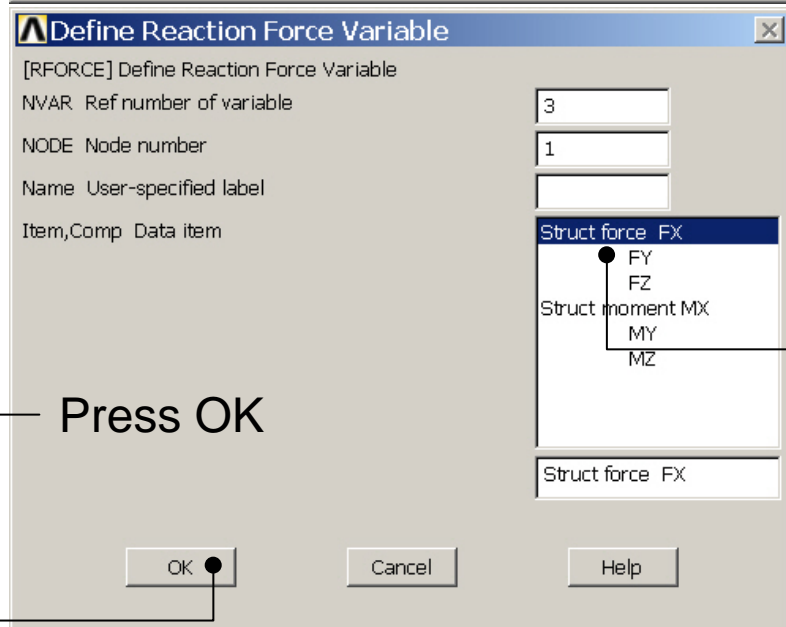
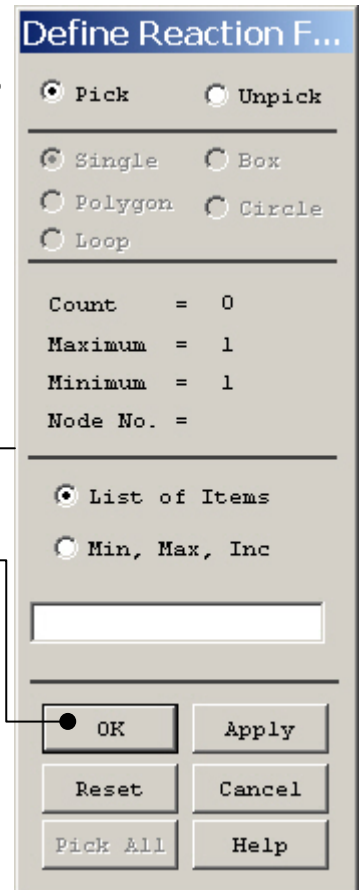
Example – Add Time-History Var.



Pick the bottom node

Select Reaction forces

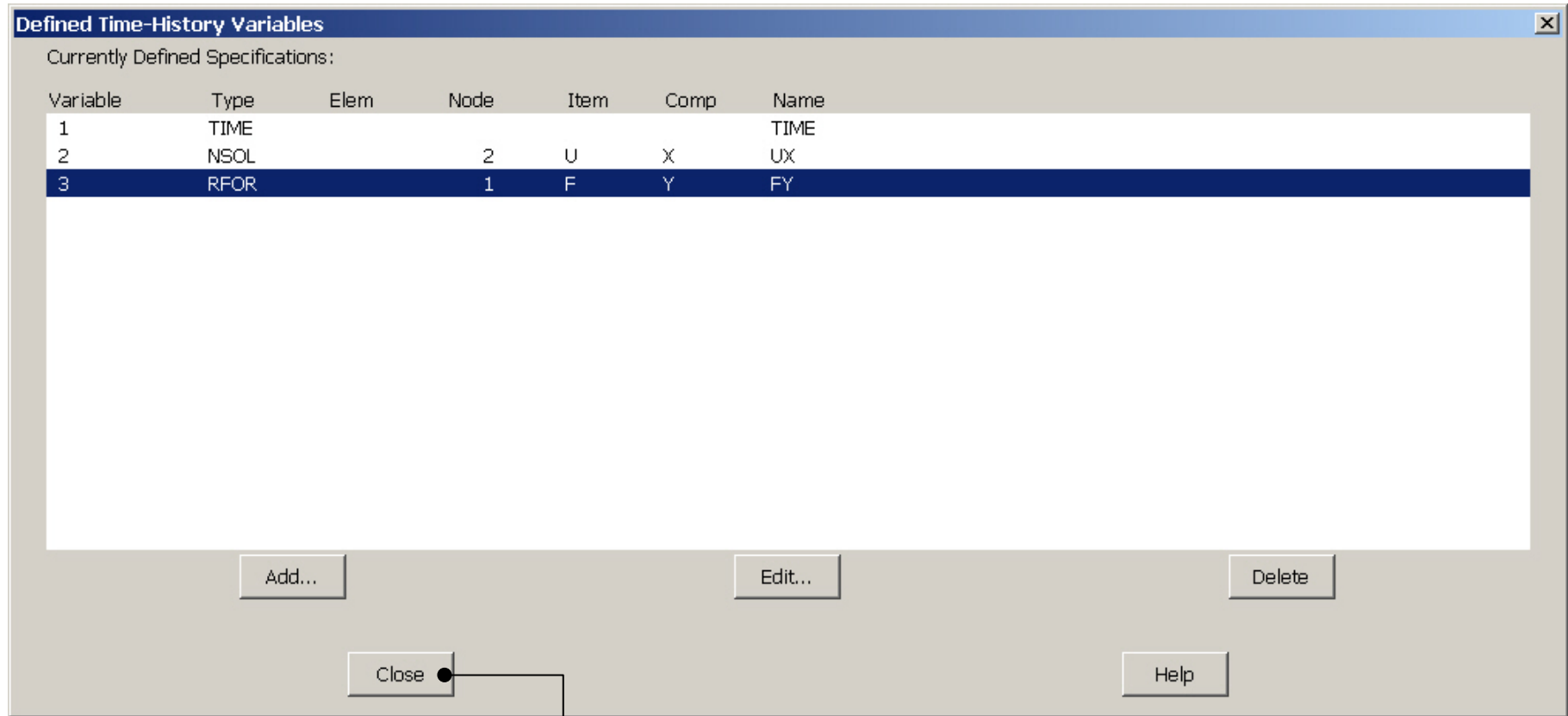
Press OK



Select Struct force FY

Example0510

Example – Add Time-History Var.



Press Close

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)

Buttons at the bottom: OK, Apply, Cancel, Help.

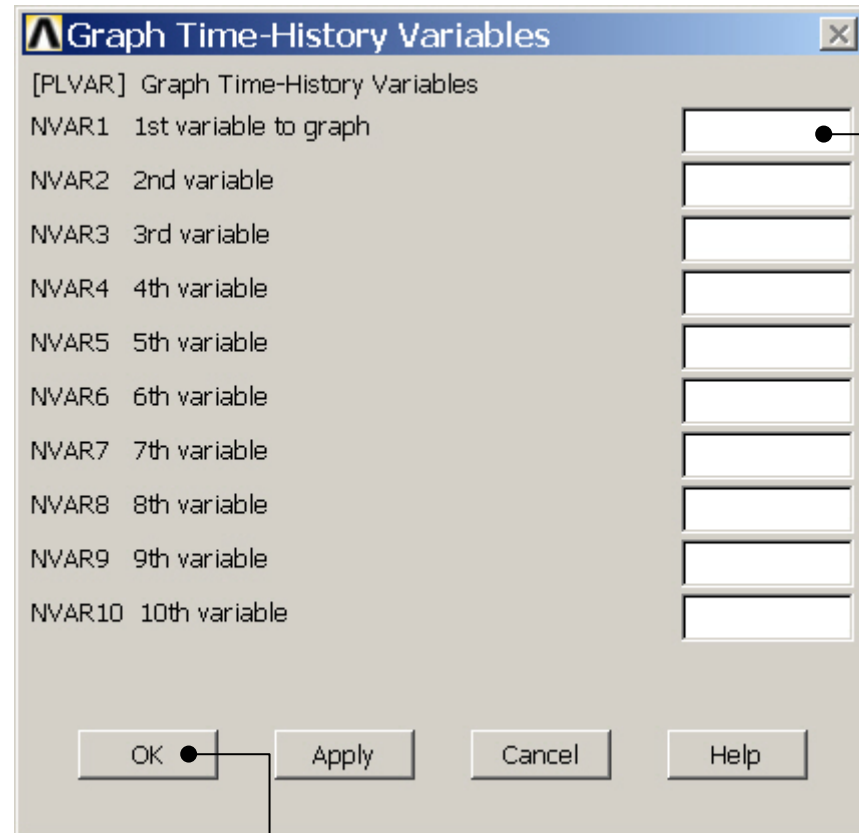
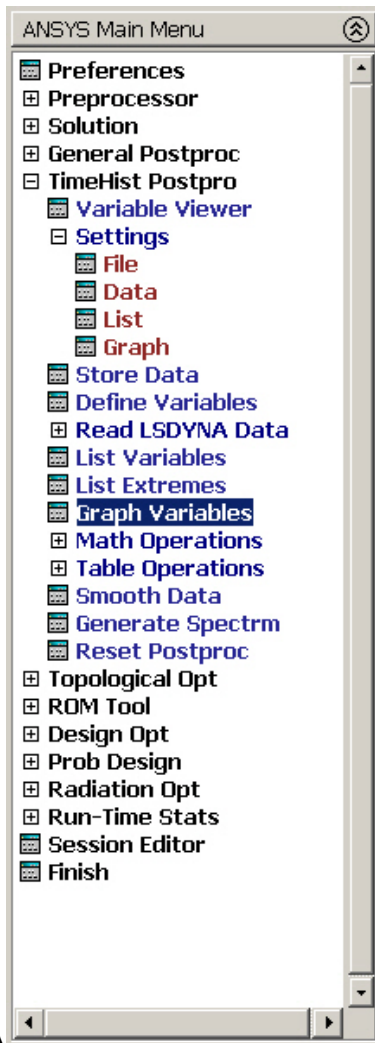
Annotations on the right side:

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

Example0510

ANSYS Computational Mechanics, AVO, LSpjerg

Example – Graph Variables

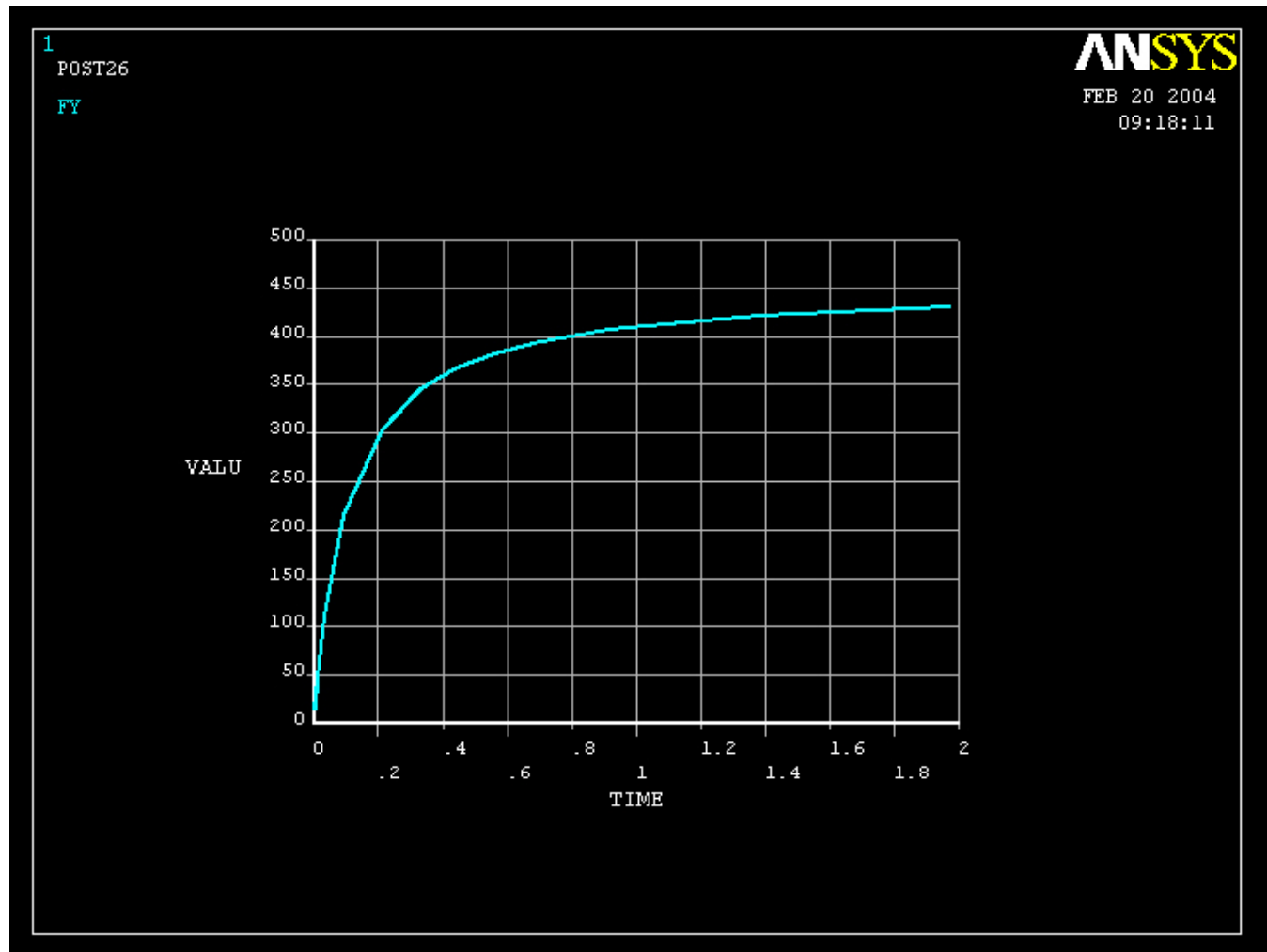


Enter 3 to plot the reaction force FY on the Y-axis

Press OK

Example0510

Example – Graph Variables



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 'OK' button is highlighted.

Enter Deformation UX

Enter Force FY

Press OK

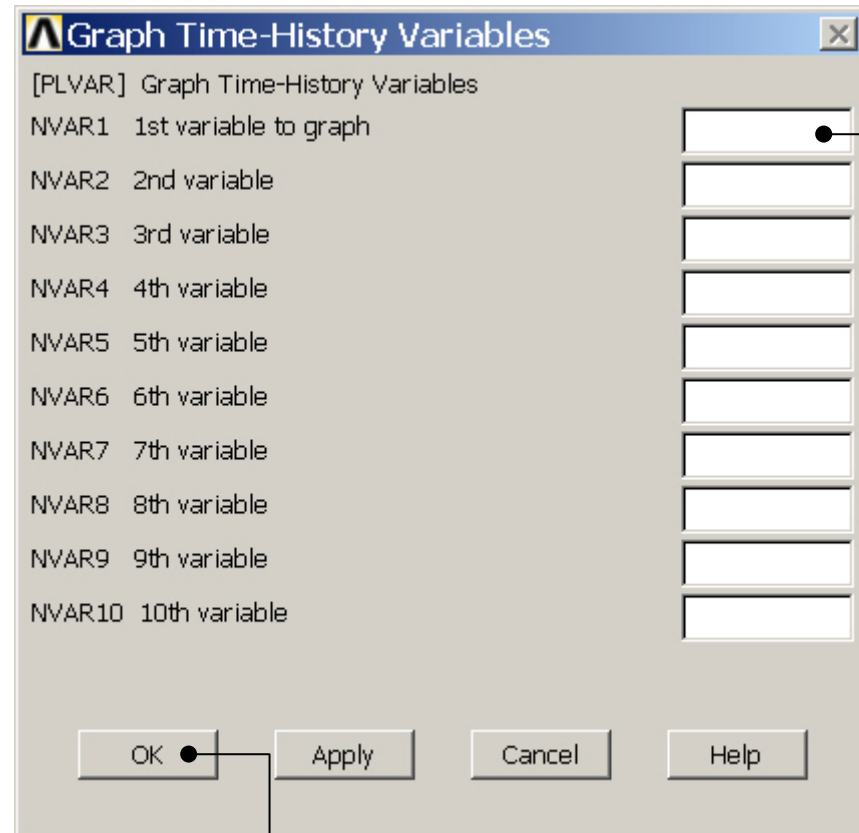
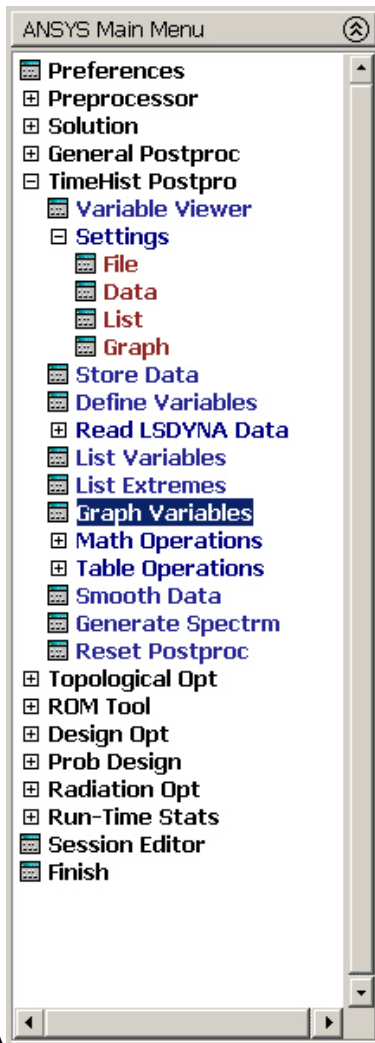
ANSYS

Computational Mechanics, AAU, Esbjerg

Example0510

27

Example – Graph Variables



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

Press OK

Example0510

Example - Graph

