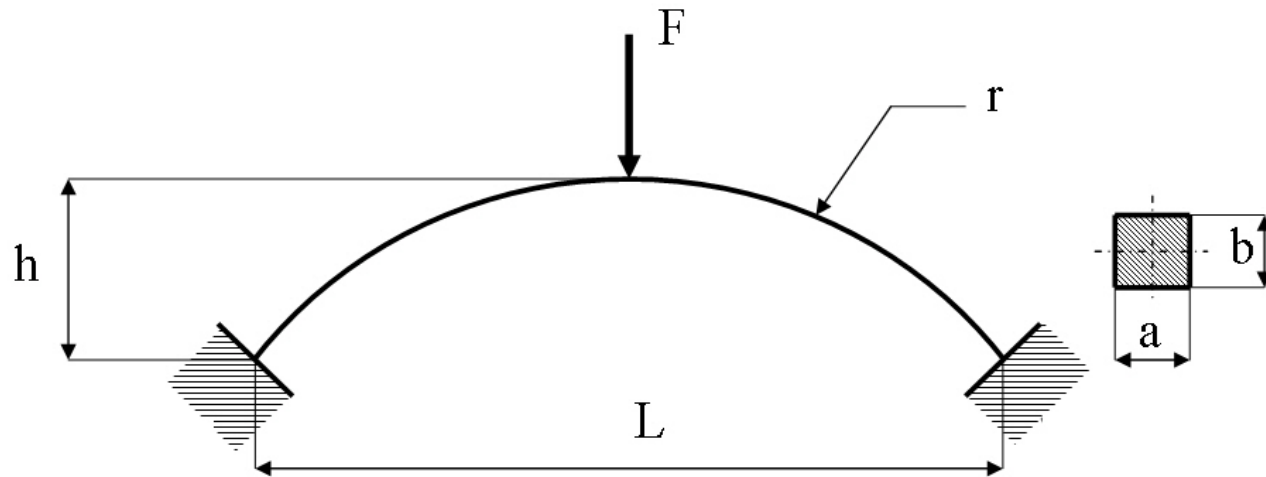


Course in ANSYS

Example0530

Example – Arch



Objective:

Plot the P-U curve for the nonlinear behaviour

Tasks:

Model the geometry

Run a static linear analysis

Run the nonlinear analysis

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

$$\nu = 0.3$$

$$L = 2000\text{mm}$$

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

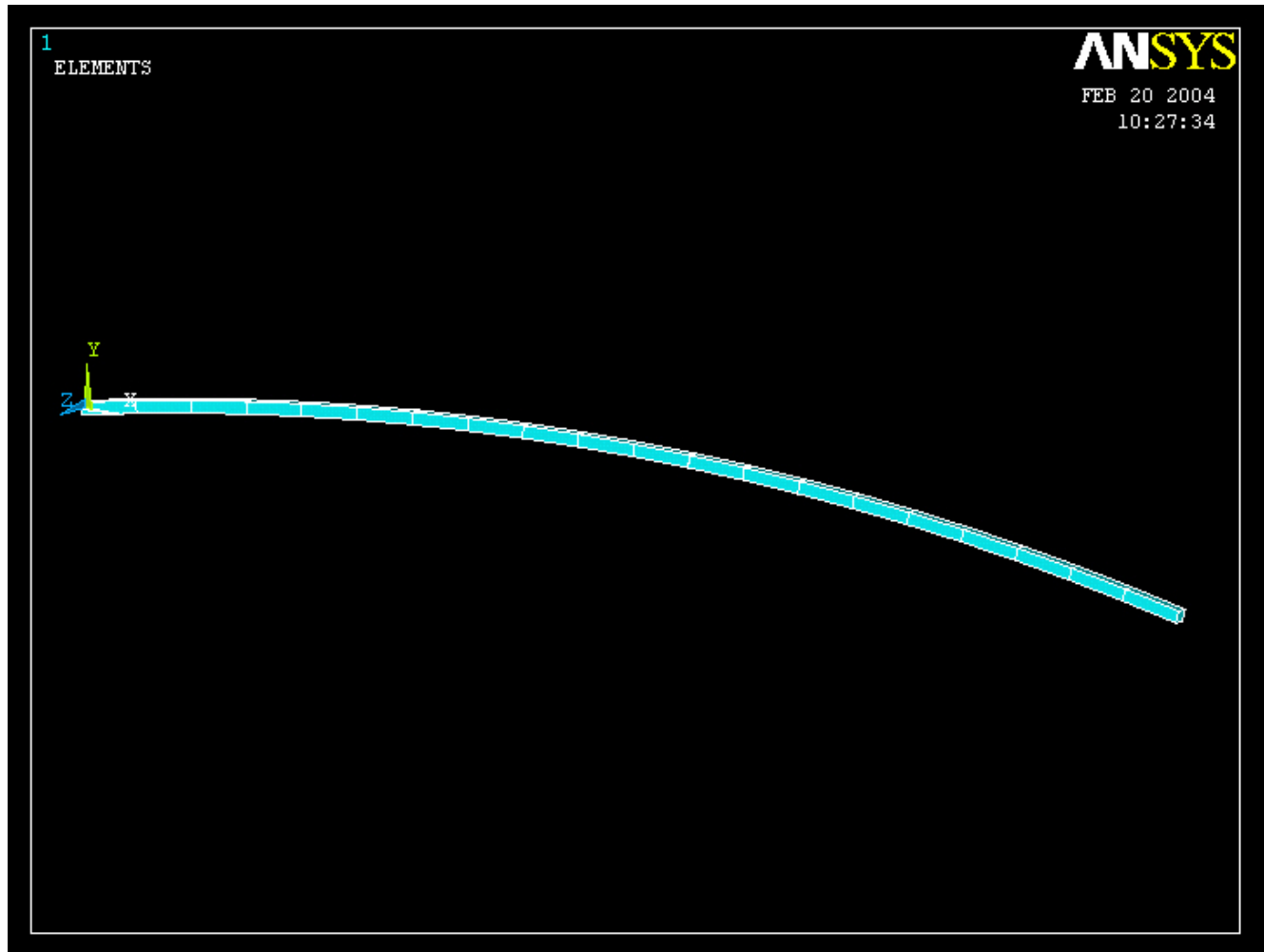
$$b = 20\text{mm}$$

$$h = 100\text{mm in } L/2$$

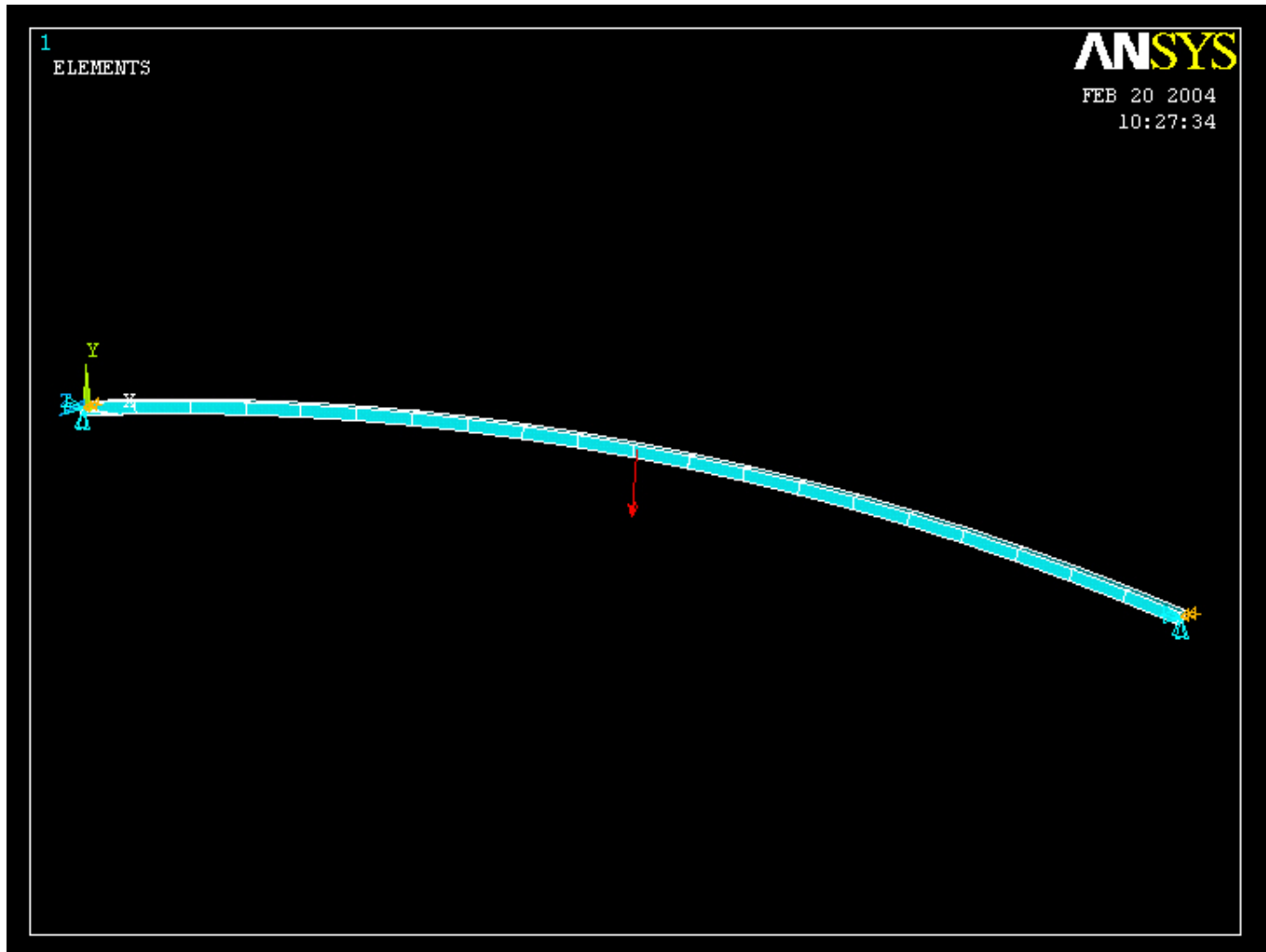
$$I = 13333.33\text{mm}^4$$

$$F = 10000\text{N}$$

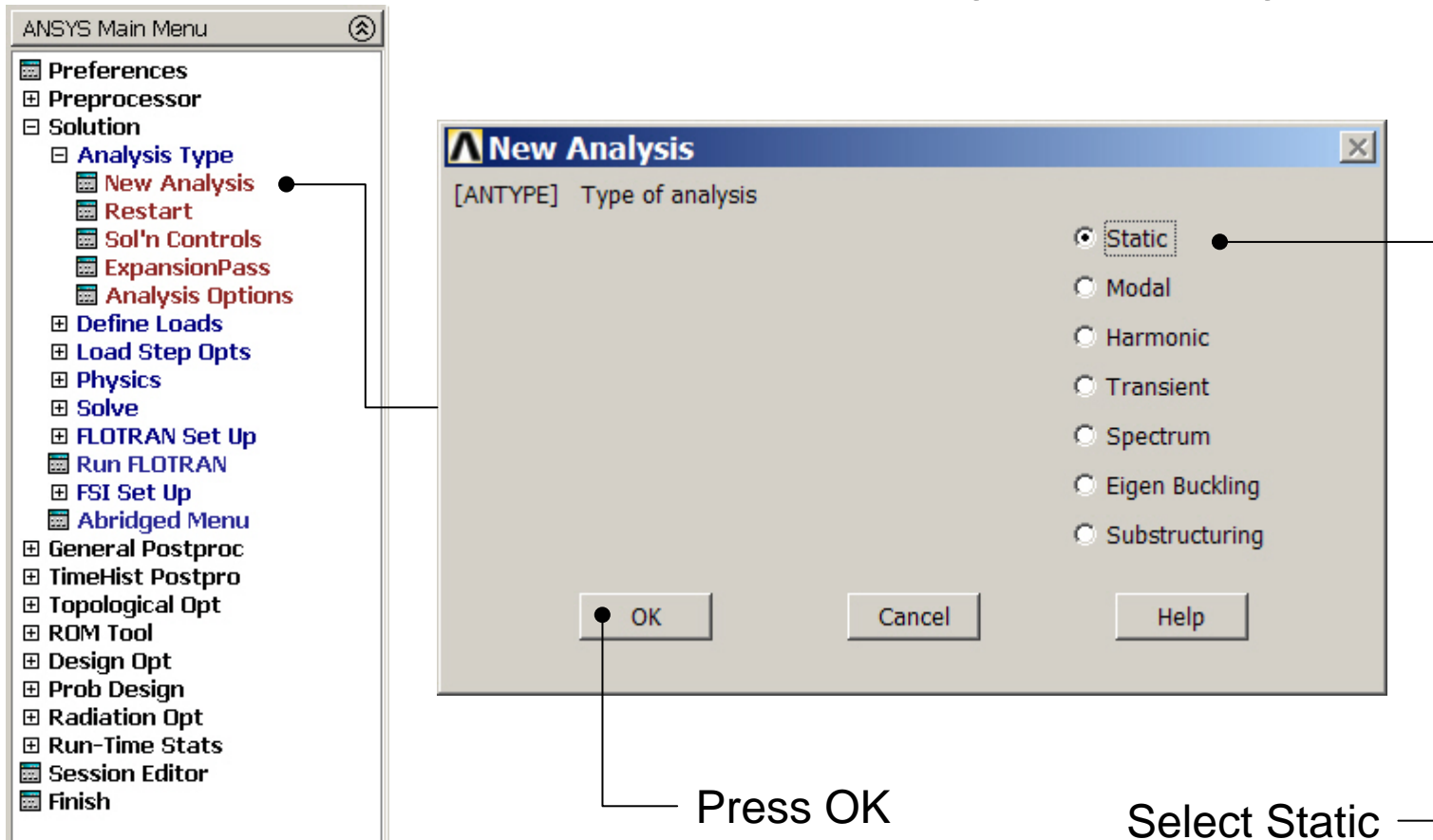
Example - Arch



Example - Arch

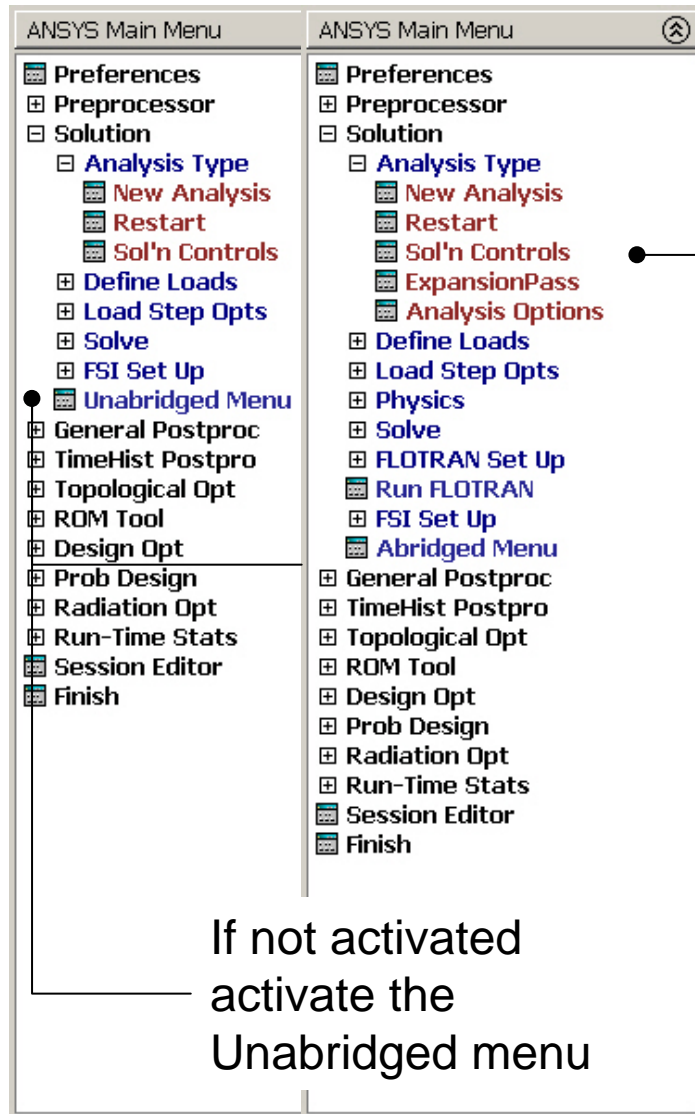


Example – Analysis Type



Example0530

Static solution – Analysis Options

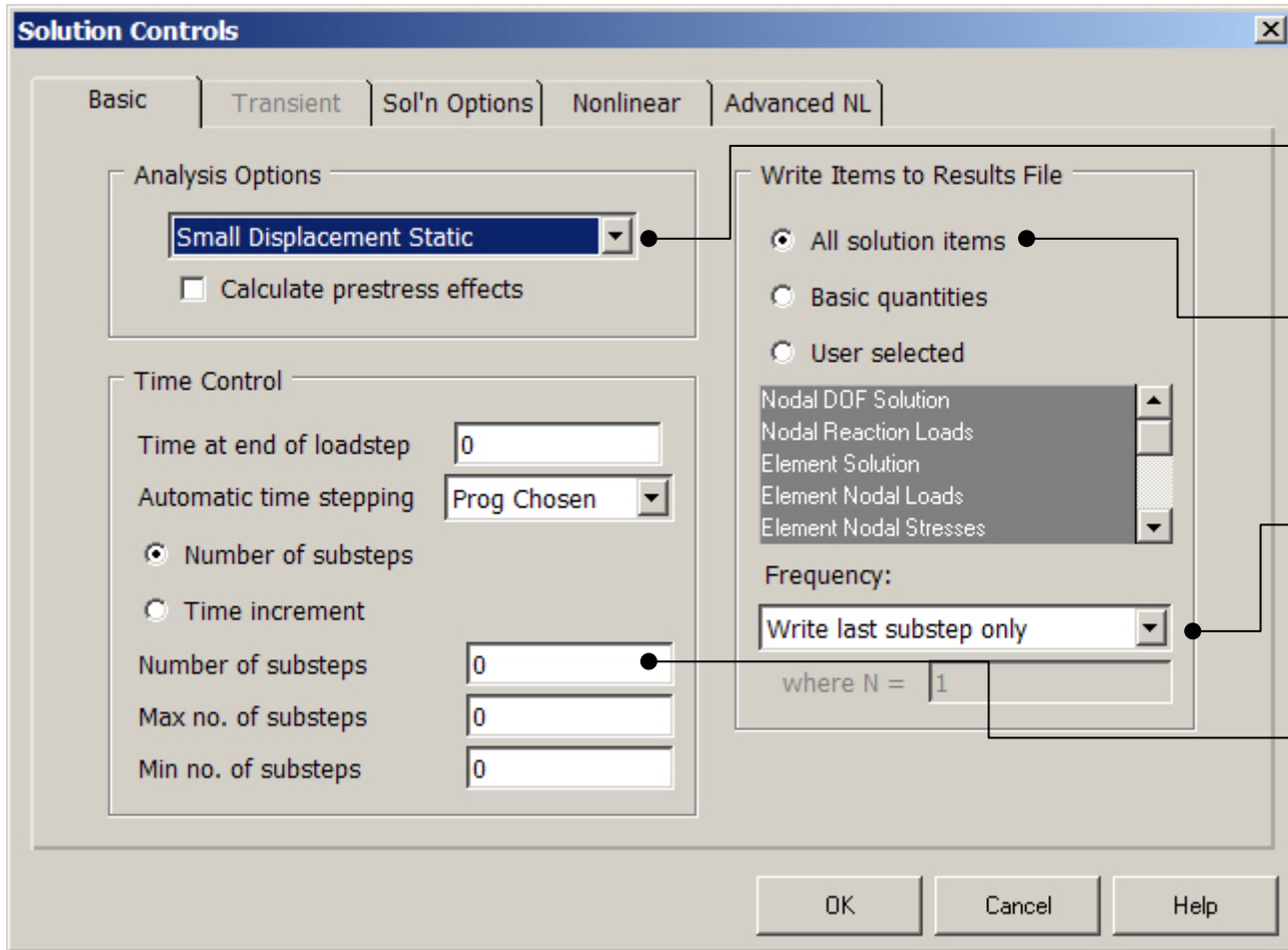


Select Sol'n Controls

If not activated
activate the
Unabridged menu

Example0530

Example – Solution Controls

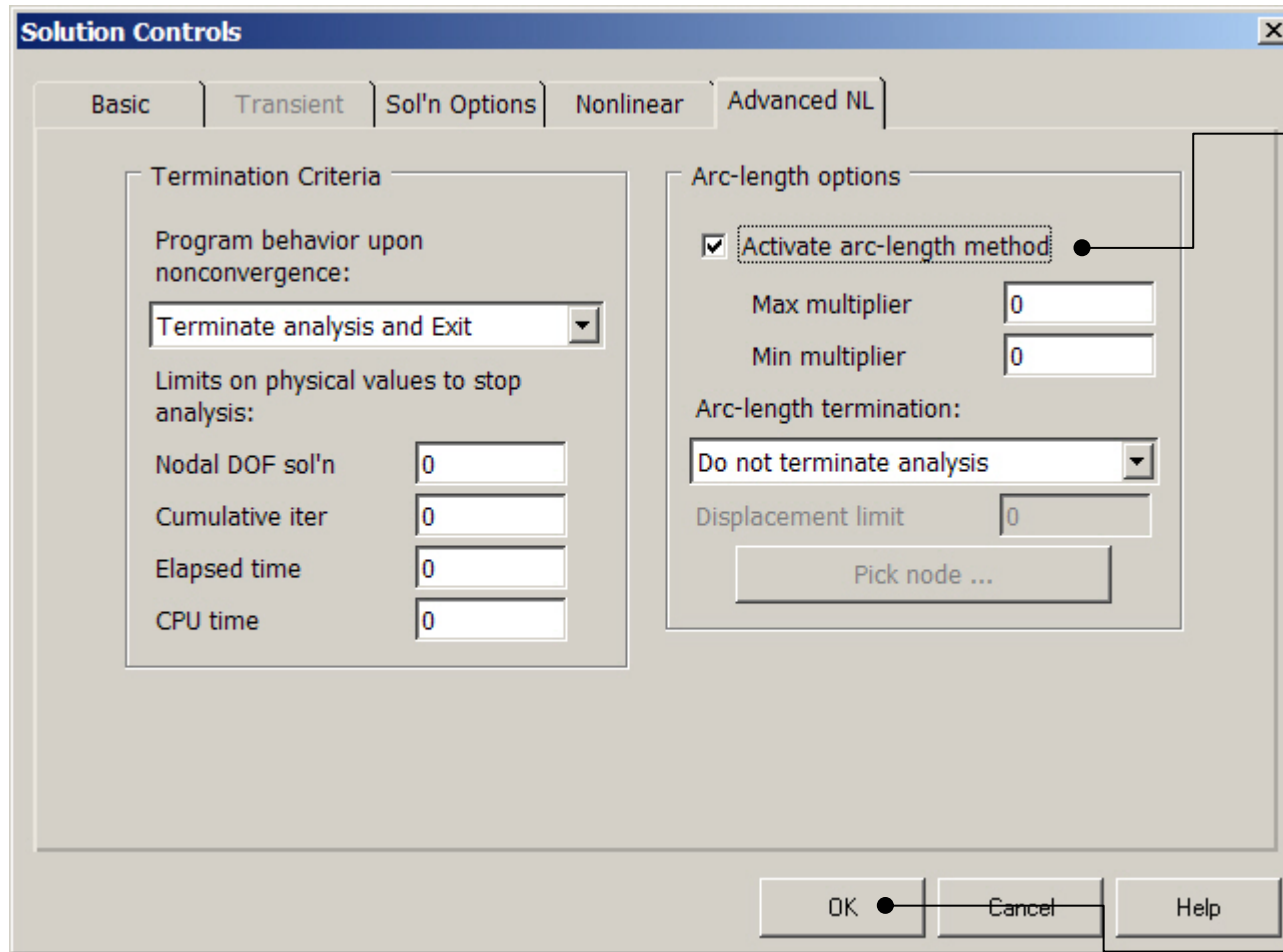


The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Static' selected in the dropdown menu. The 'Time Control' section has 'Time at end of loadstep' set to 0, 'Automatic time stepping' set to 'Prog Chosen', and 'Number 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.

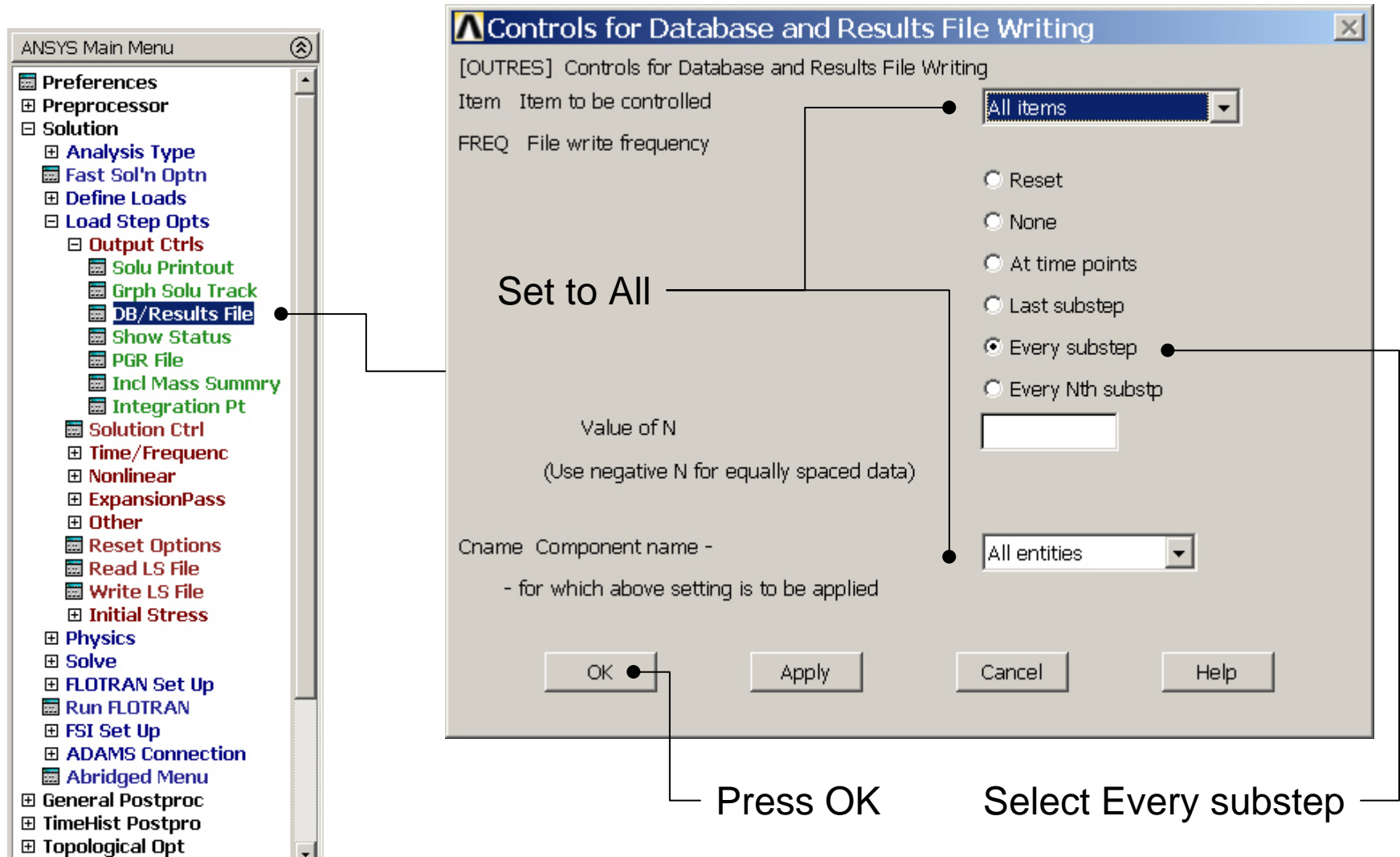
Example – Solution Controls



Activate the arc-length method

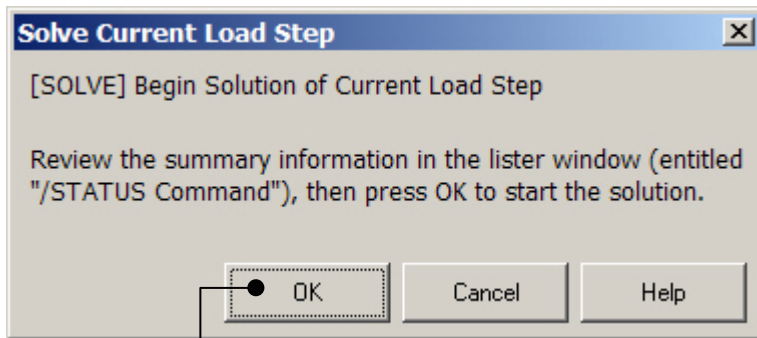
Press OK

Example – Output Ctrls



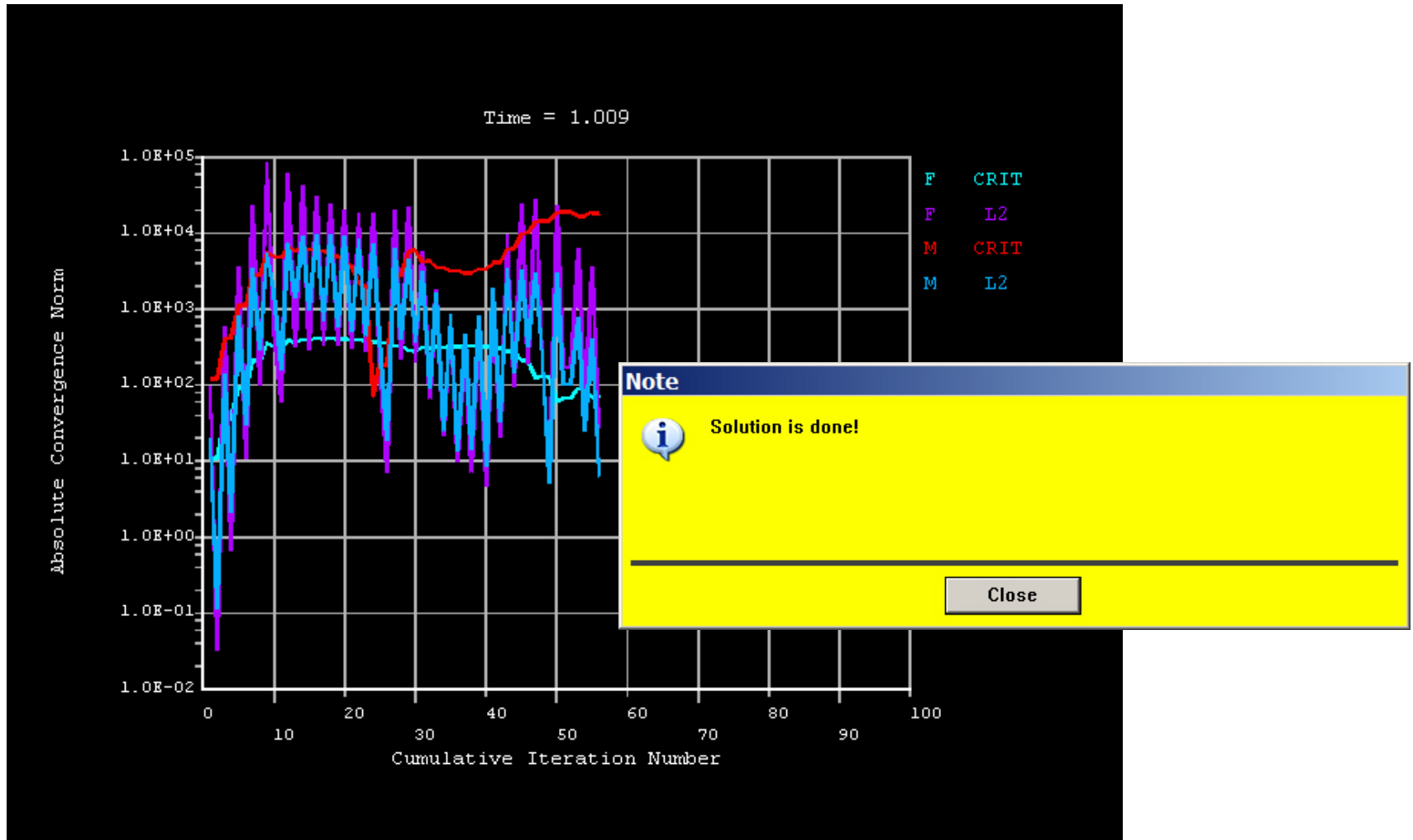
Example - Solve

Solution > Solve > Current LS

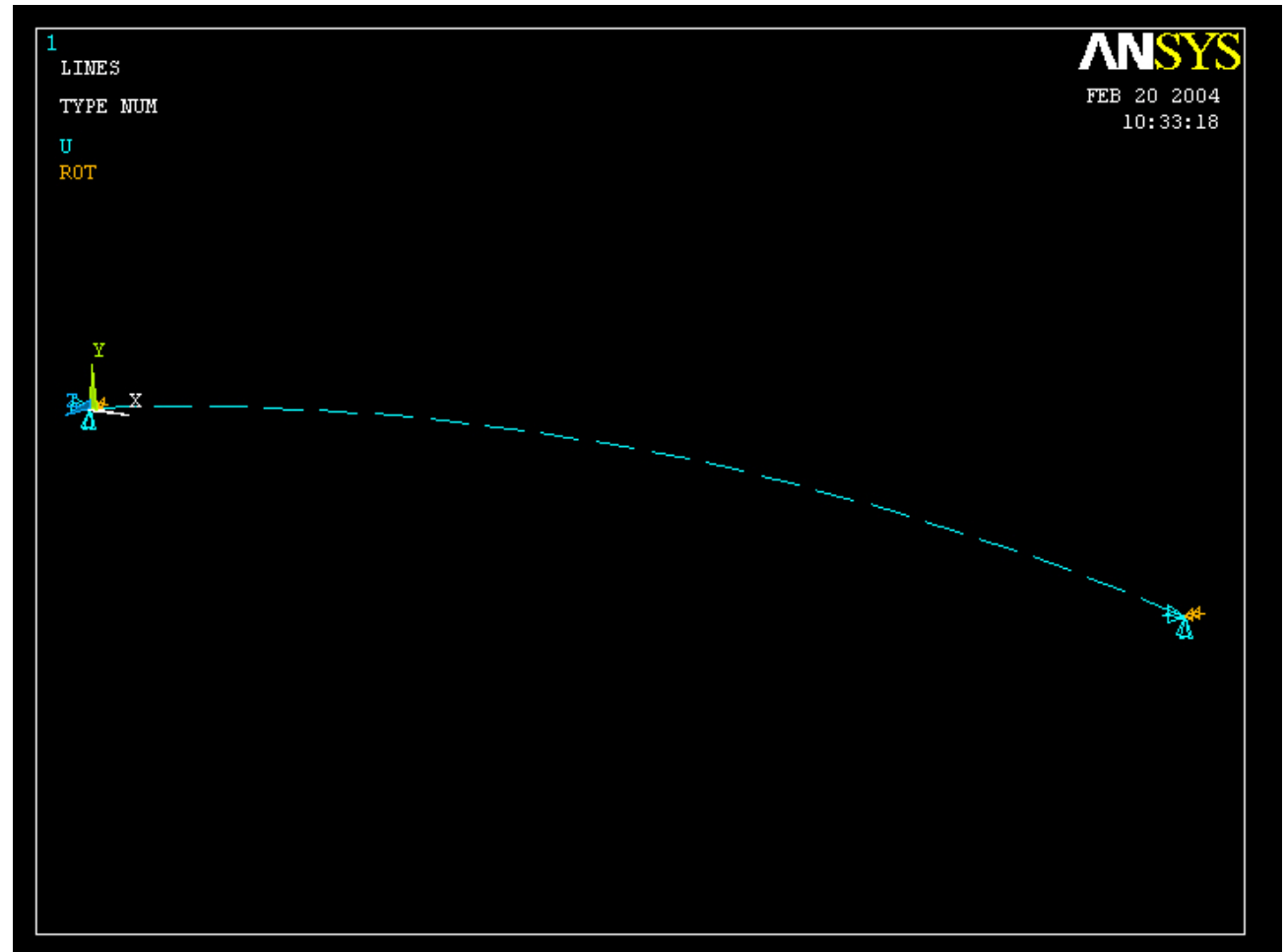
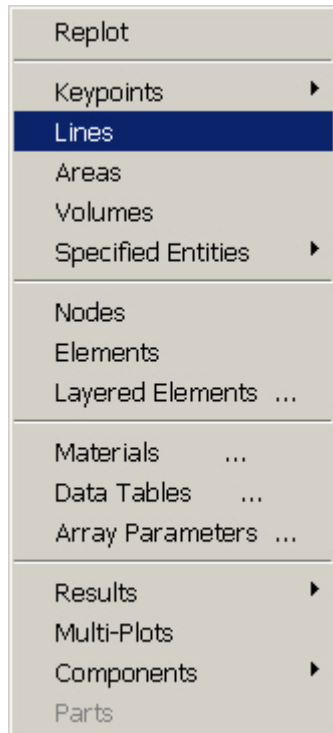


Press OK

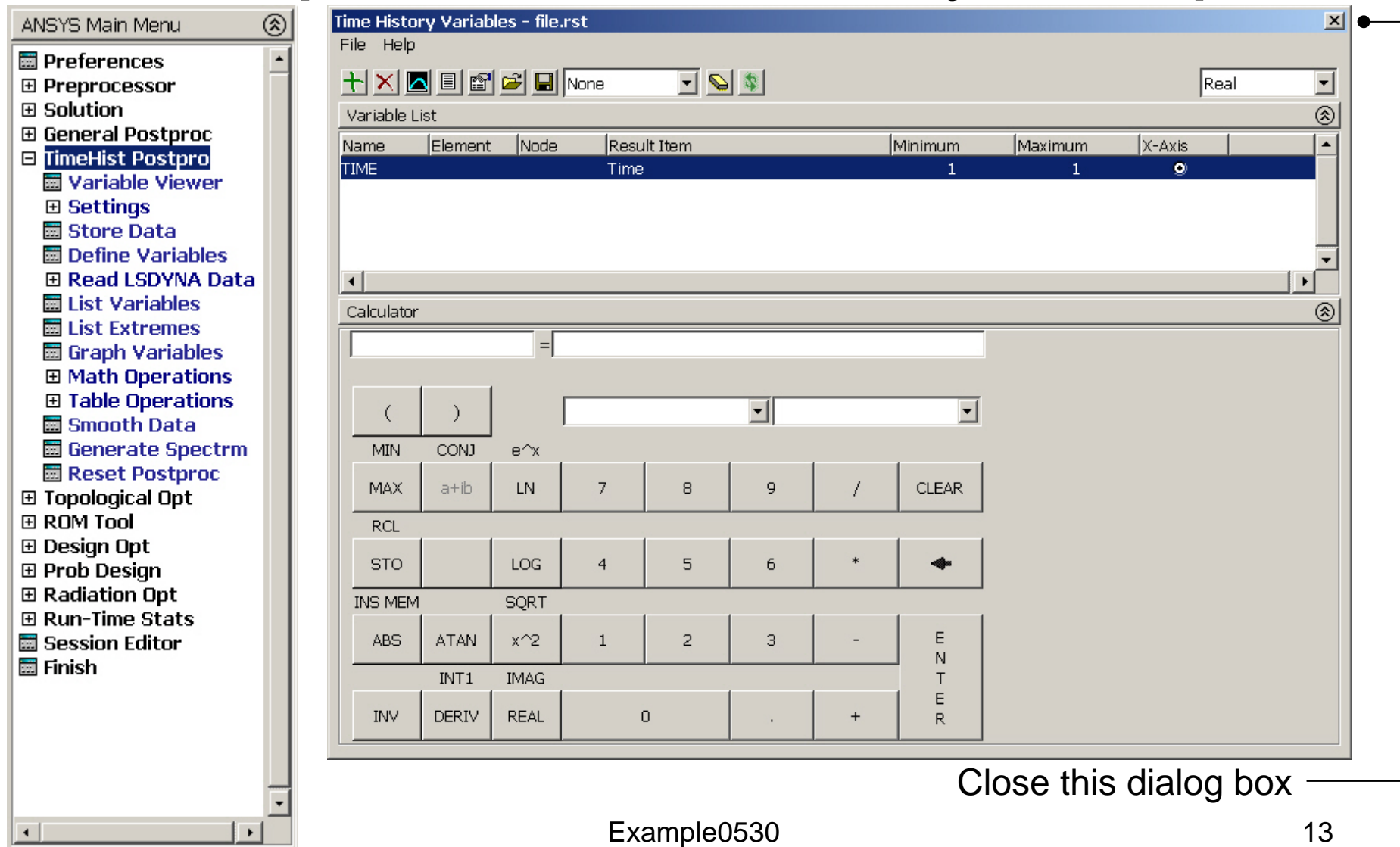
Example - Convergence



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	<input checked="" type="radio"/>

Calculator

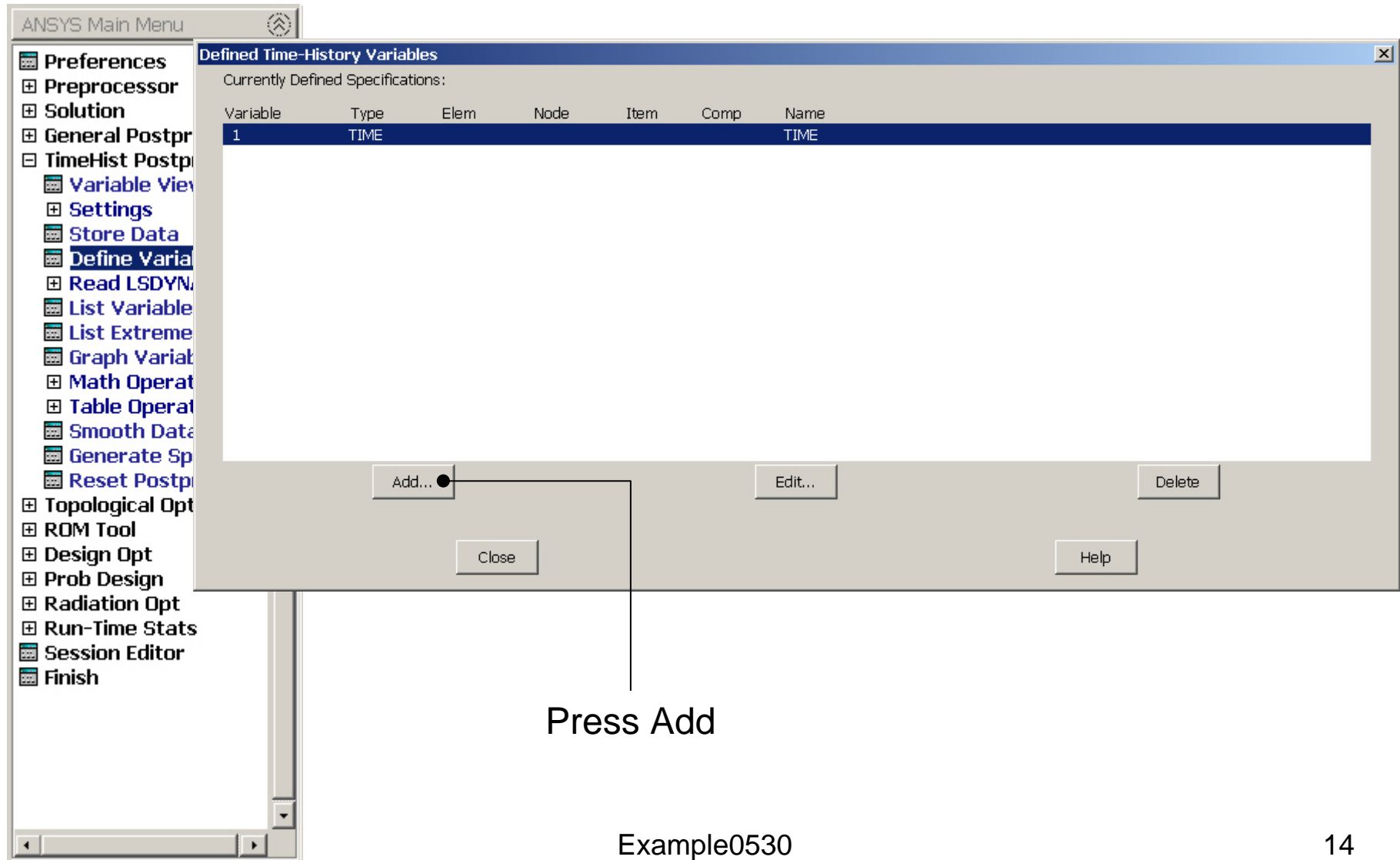
Calculator interface with buttons for MIN, MAX, RCL, STO, INS MEM, SQRT, INT1, IMAG, INV, DERIV, REAL, and a numeric keypad.

Close this dialog box

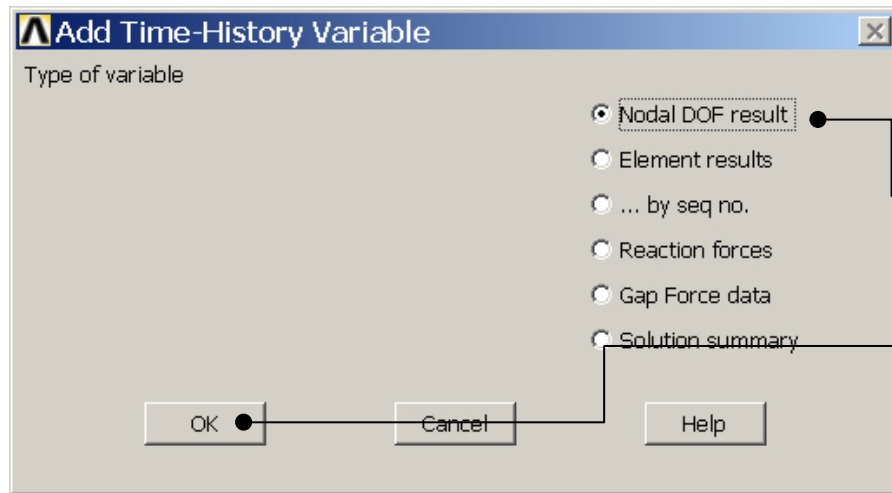
Example0530

13

Example – Define Variables



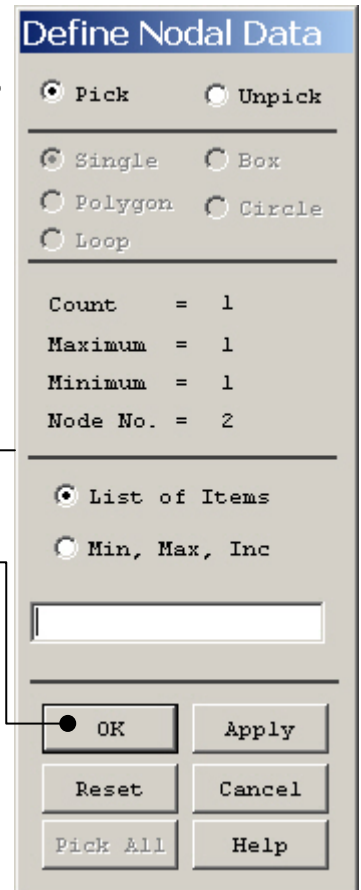
Example – Add Time-History Var.



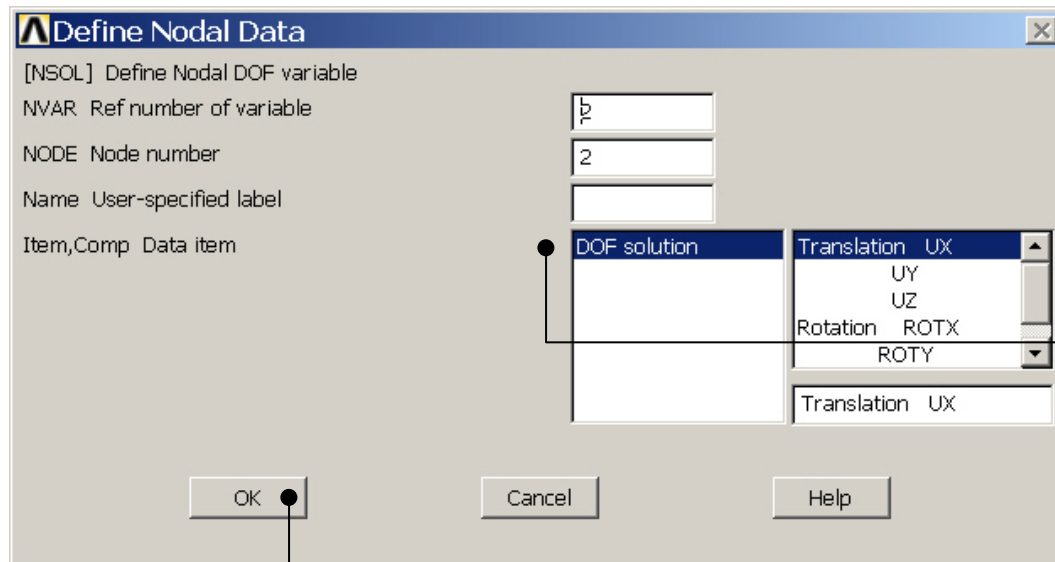
Pick the middle node

Select Nodal DOF result

Press OK

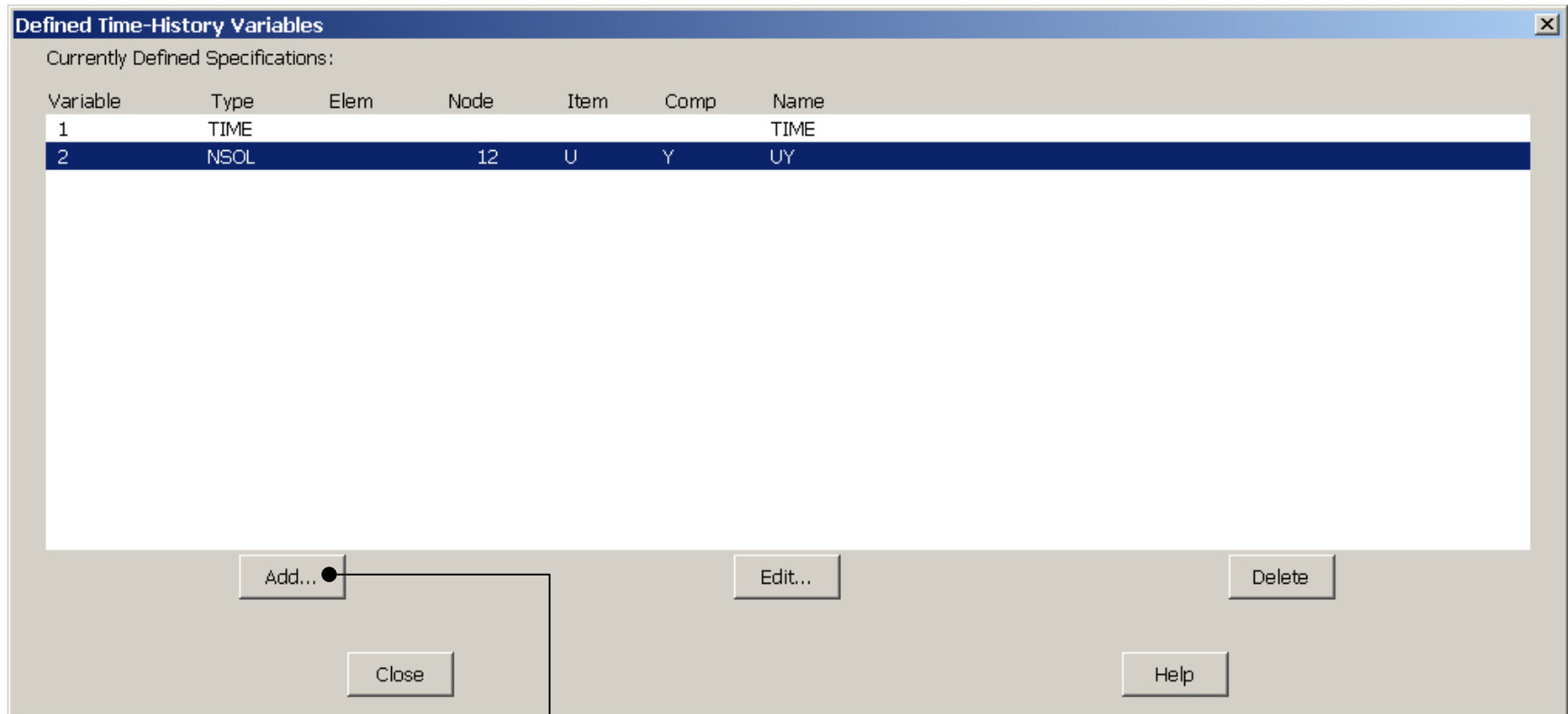


Press OK



Select DOF solution and Translation UY

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 UY for the top node on the X-axis (points to the Single variable no. field)
- Press OK (points to the OK button)

Example0530

ANSYS Computational Mechanics, AAO, LSpjerg

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', and the 'Y-axis range' is set to 'Auto calculated'. The 'Specified X range' and 'Specified Y range' are both set to '1'. The 'Axis Controls' section shows 'LOGX' and 'LOGY' both 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 UY

Enter Force FY

Press OK

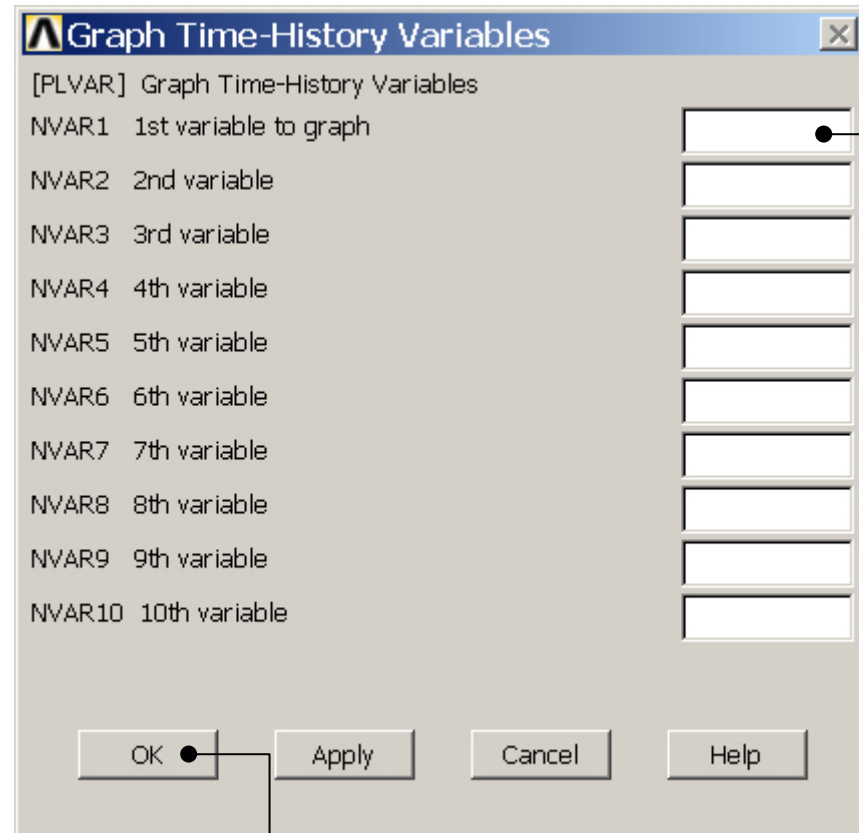
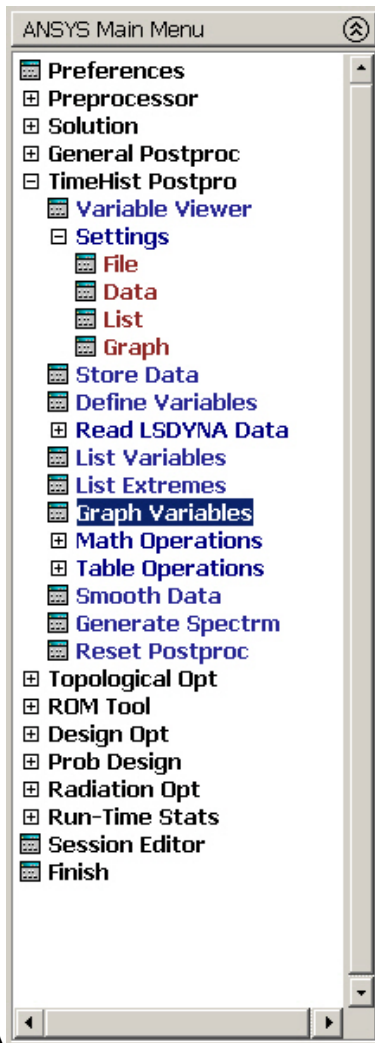
ANSYS

Computational Mechanics, AAU, Esbjerg

Example0530

18

Example – Graph Variables



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

Press OK

Example0530

Example - Graph

