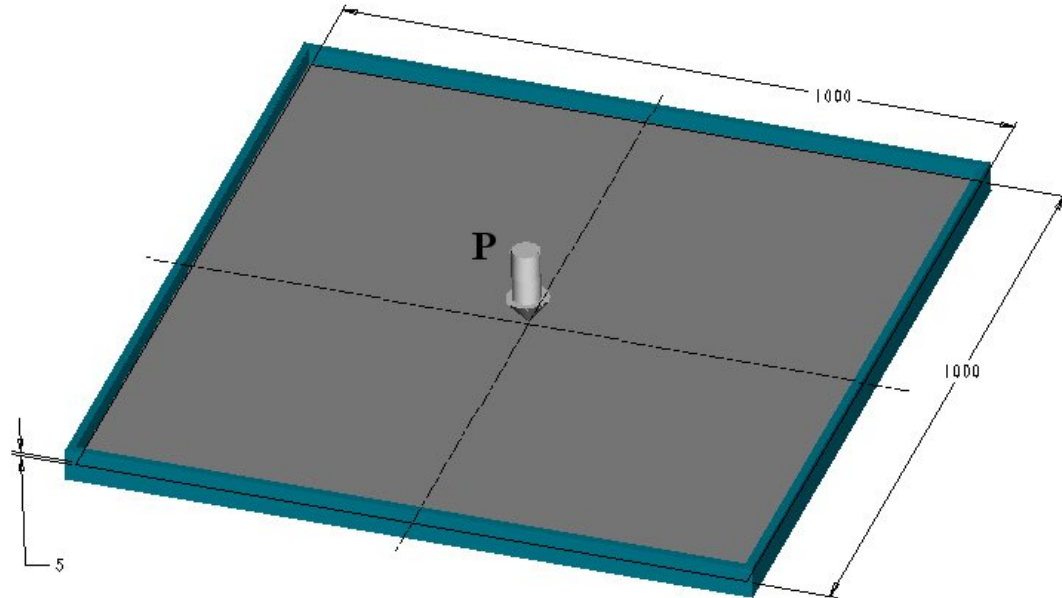


# Course in ANSYS

Example0540

# Example – Plate



## Objective:

Plot the P-U curve for the nonlinear behaviour

## Tasks:

Model the geometry

Run a static linear analysis

Run the nonlinear analysis

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

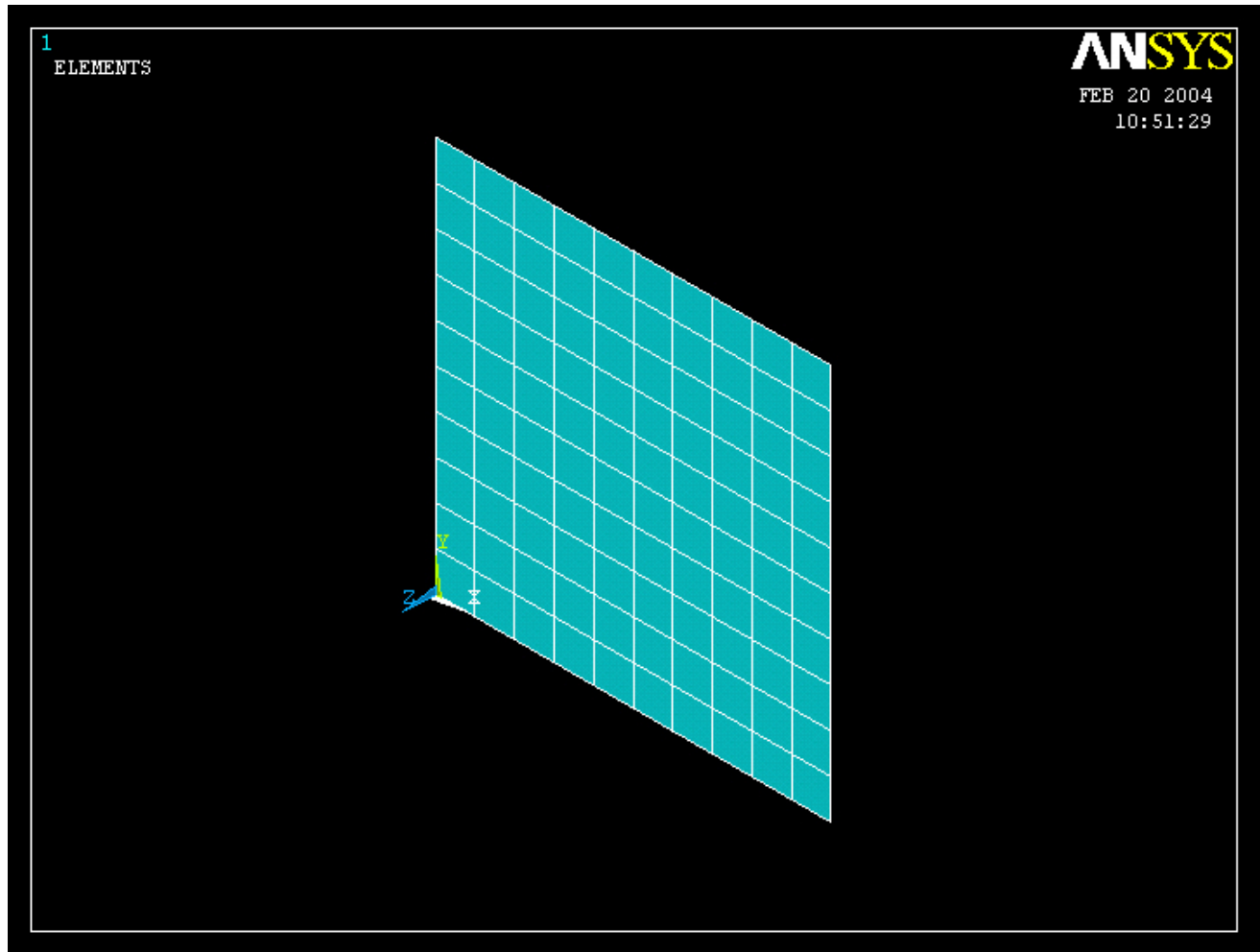
$$\nu = 0.3$$

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

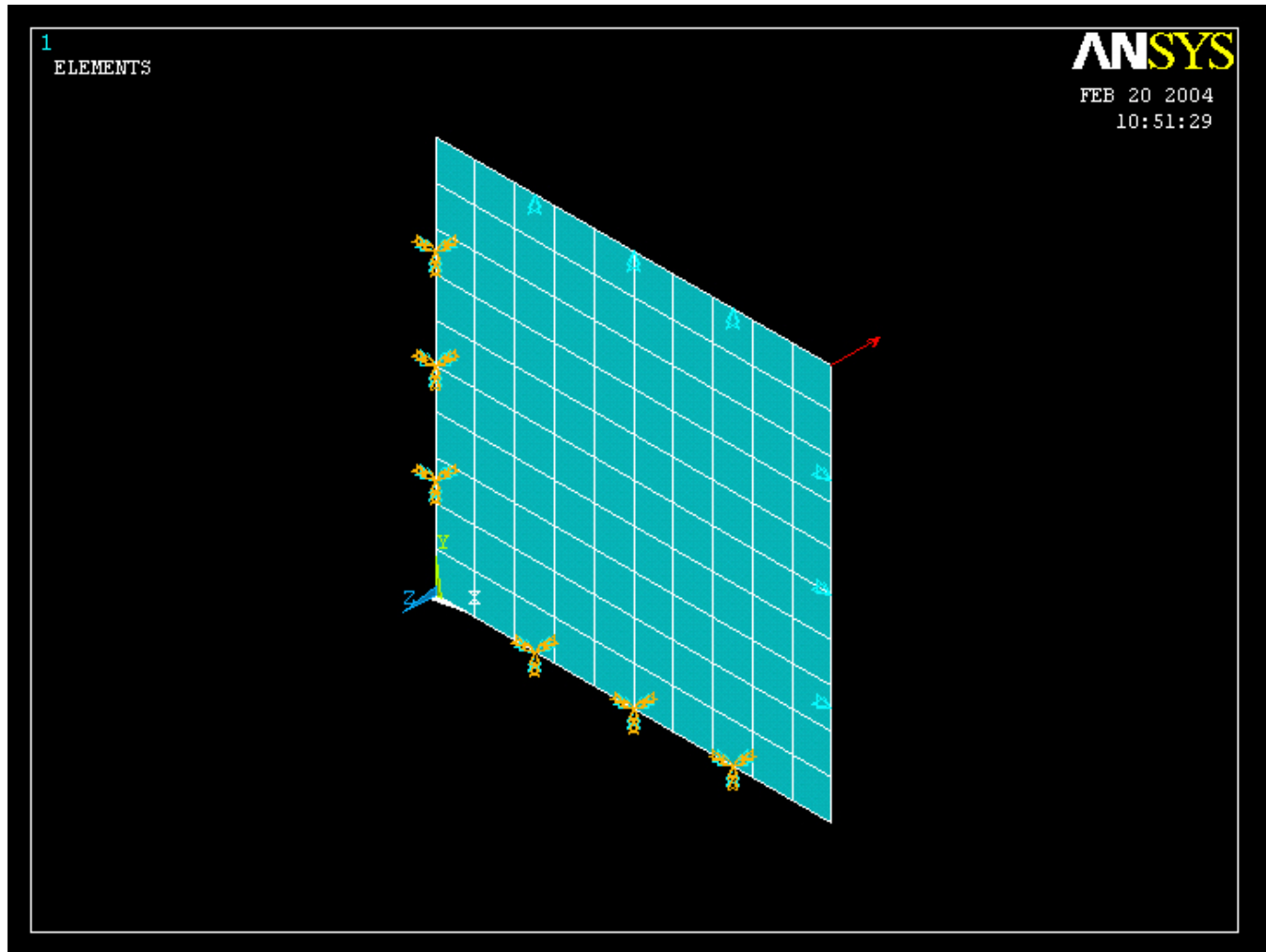
$$t = 5\text{mm}$$

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

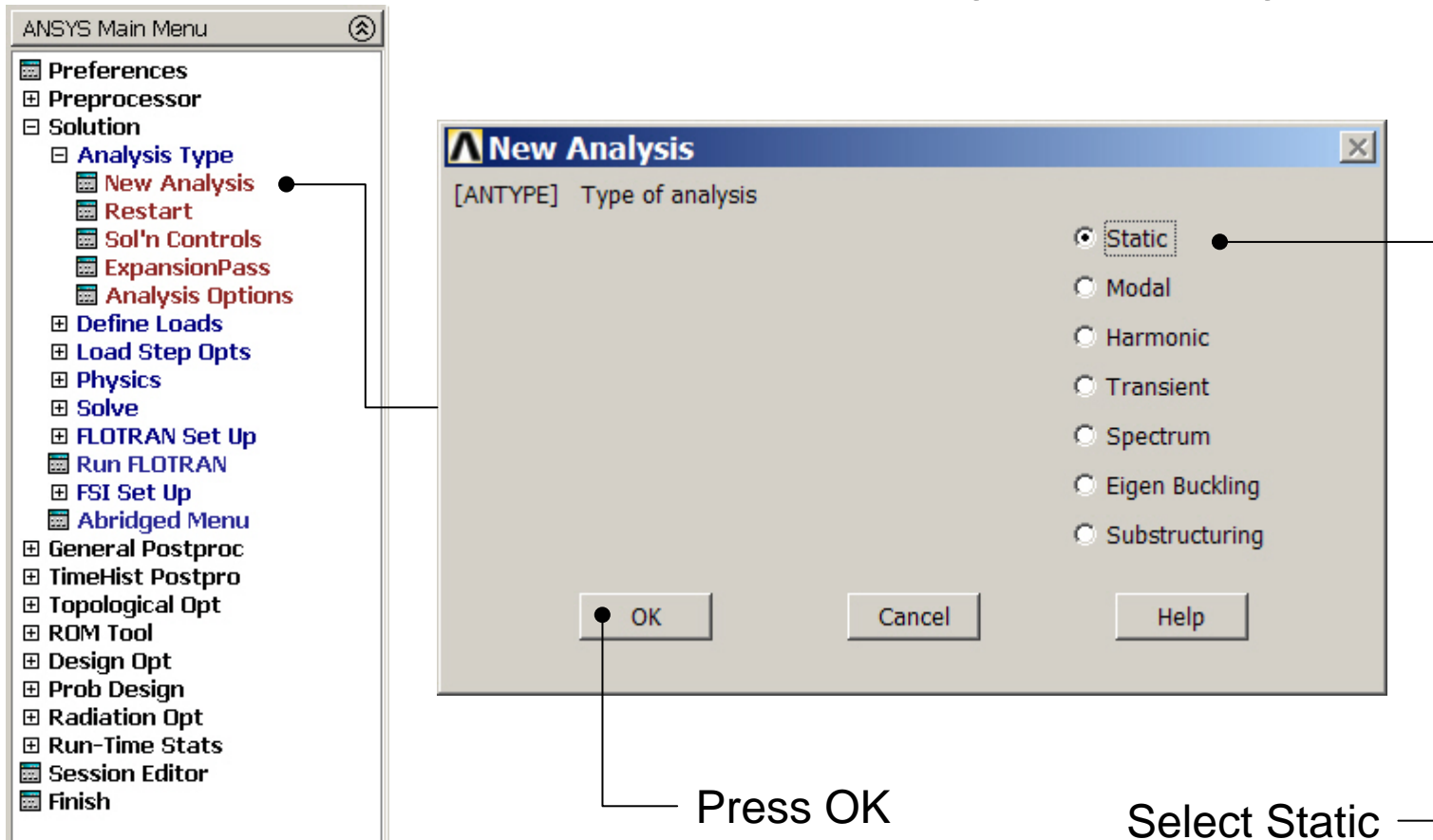
# Example - Plate



# Example - Plate

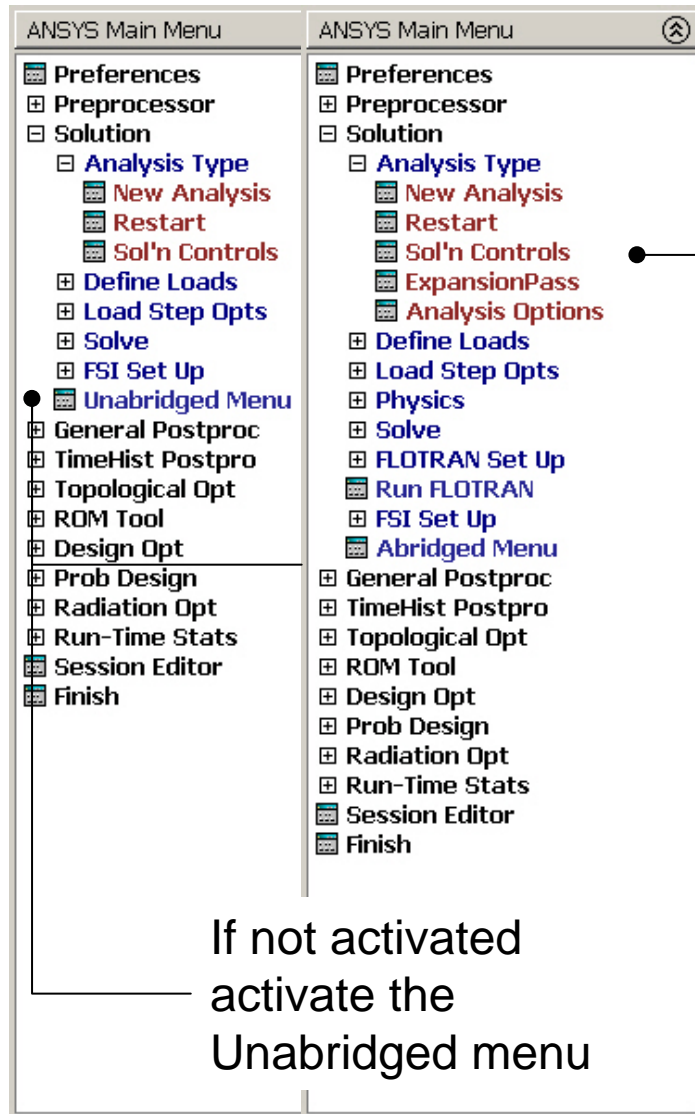


# Example – Analysis Type



Example0540

# Static solution – Analysis Options



Select Sol'n Controls

If not activated  
activate the  
Unabridged menu

Example0540

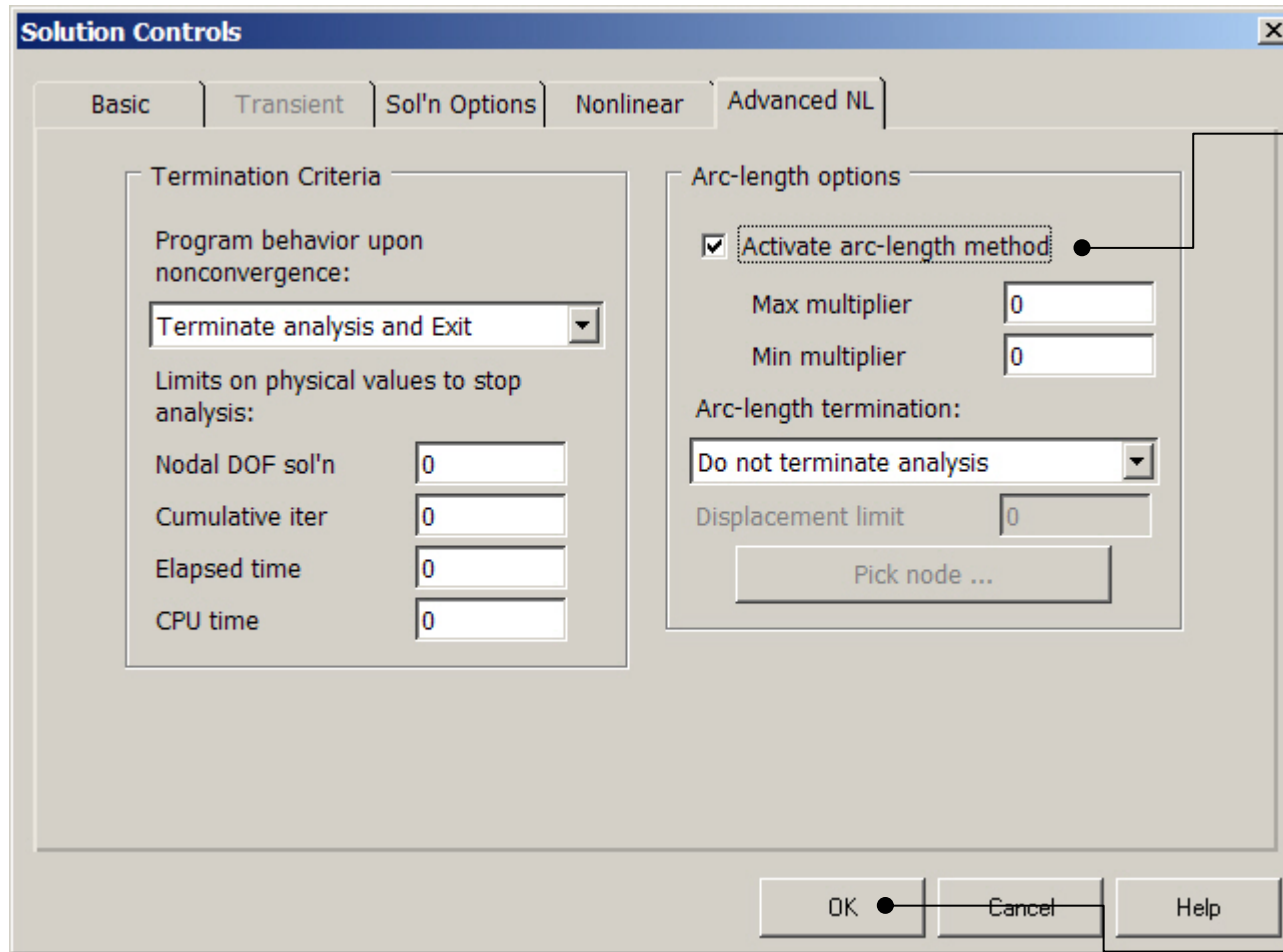
# Example – Solution Controls

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

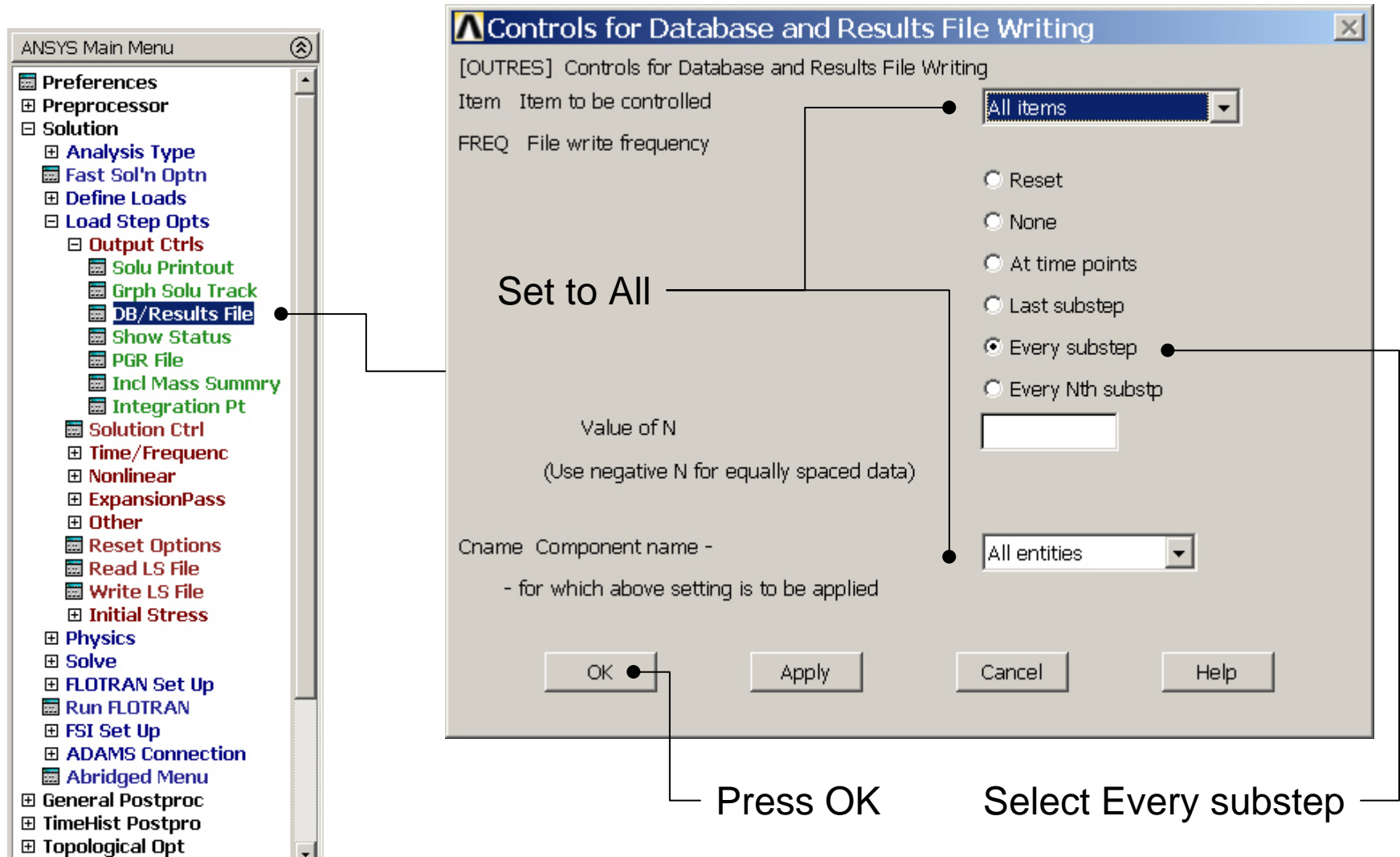


Activate the arc-length method

Press OK



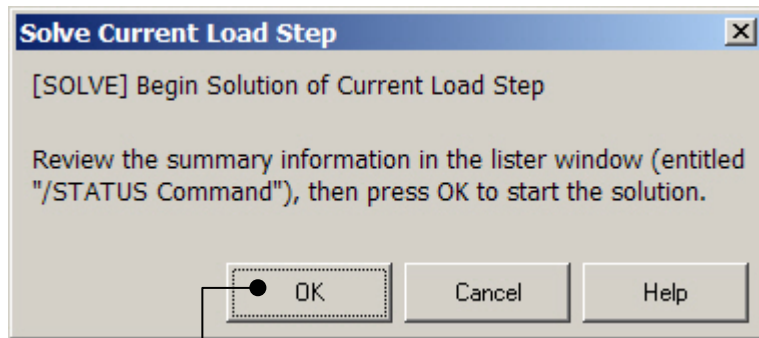
# Example – Output Ctrls



Example0540

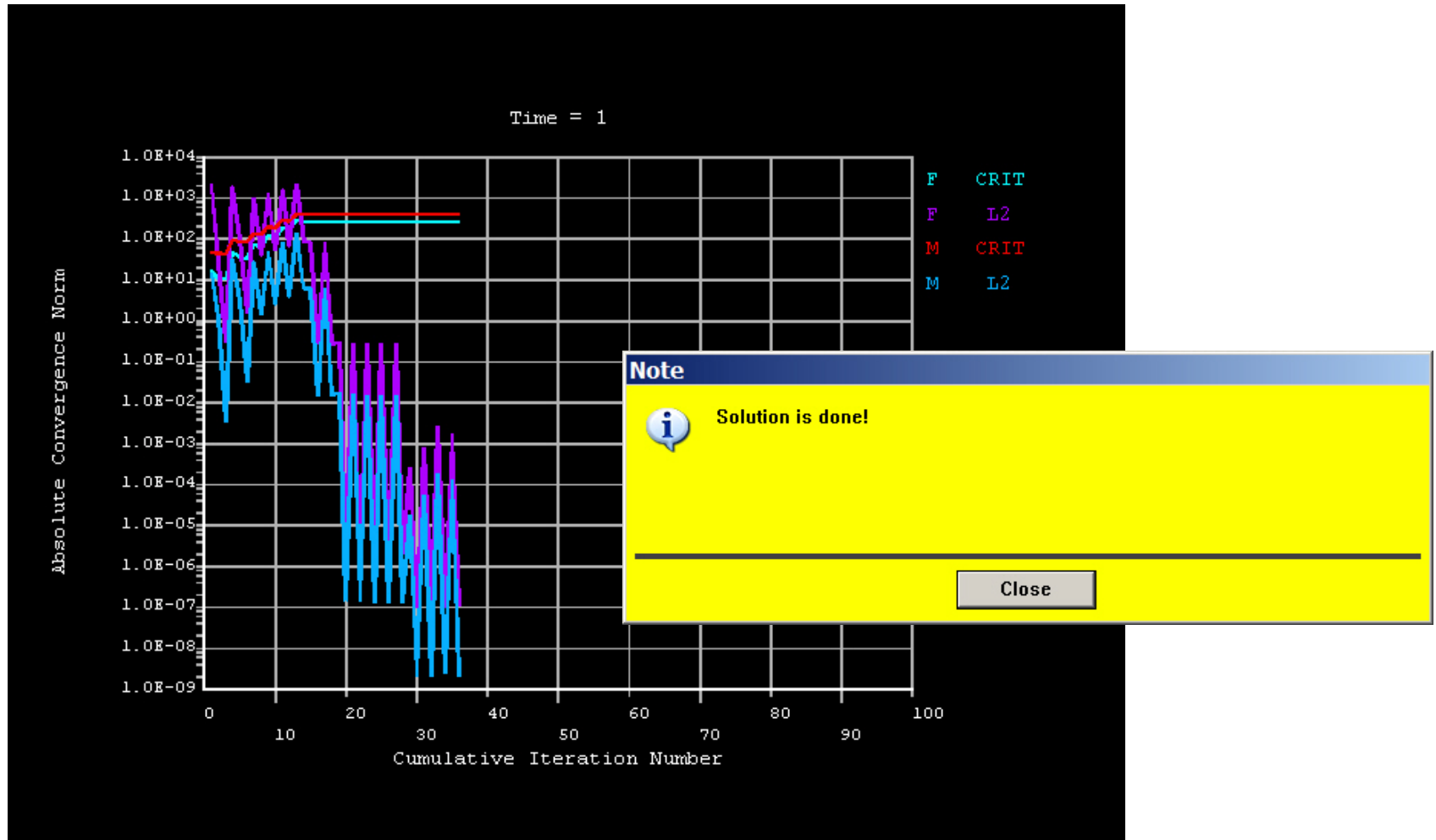
# Example - Solve

**Solution > Solve > Current LS**

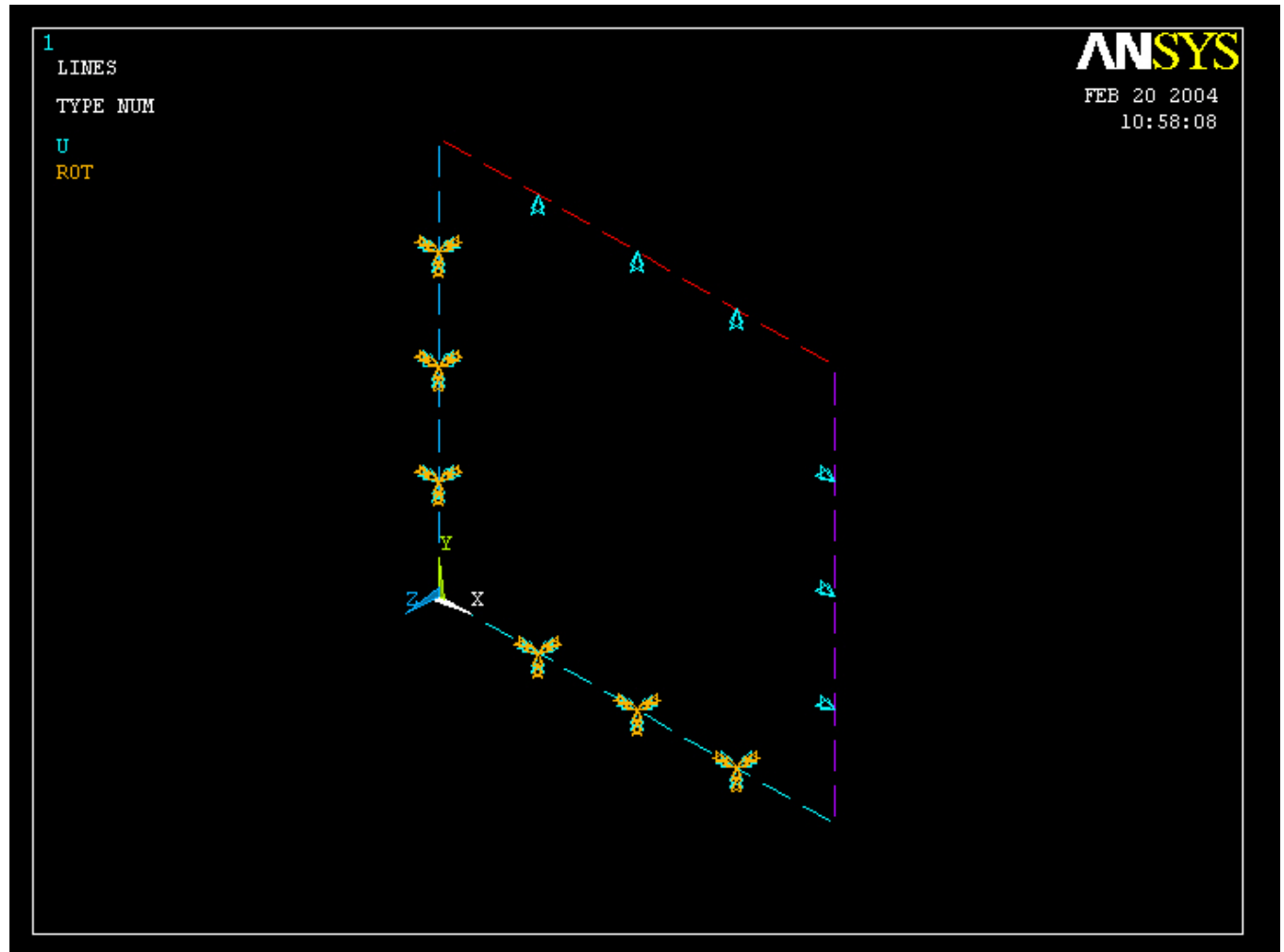
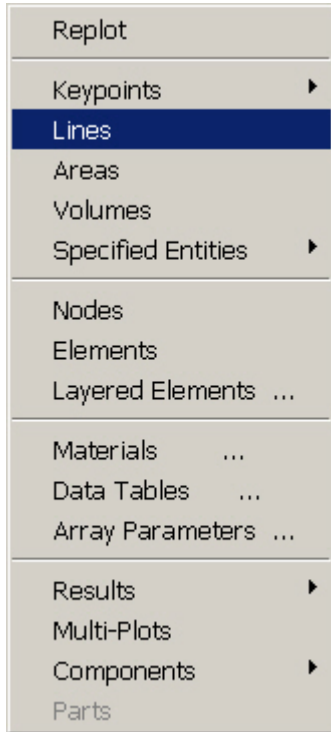


Press OK

# 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, 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, and Finish. The Time History Variables dialog box has a title bar 'Time History Variables - file.rst' and a menu bar 'File Help'. It contains a toolbar with icons for adding, deleting, and saving variables. Below the toolbar is a 'Variable List' table with columns: Name, Element, Node, Result Item, Minimum, Maximum, X-Axis, and a selection column. The table contains 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 buttons for mathematical operations and functions.

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

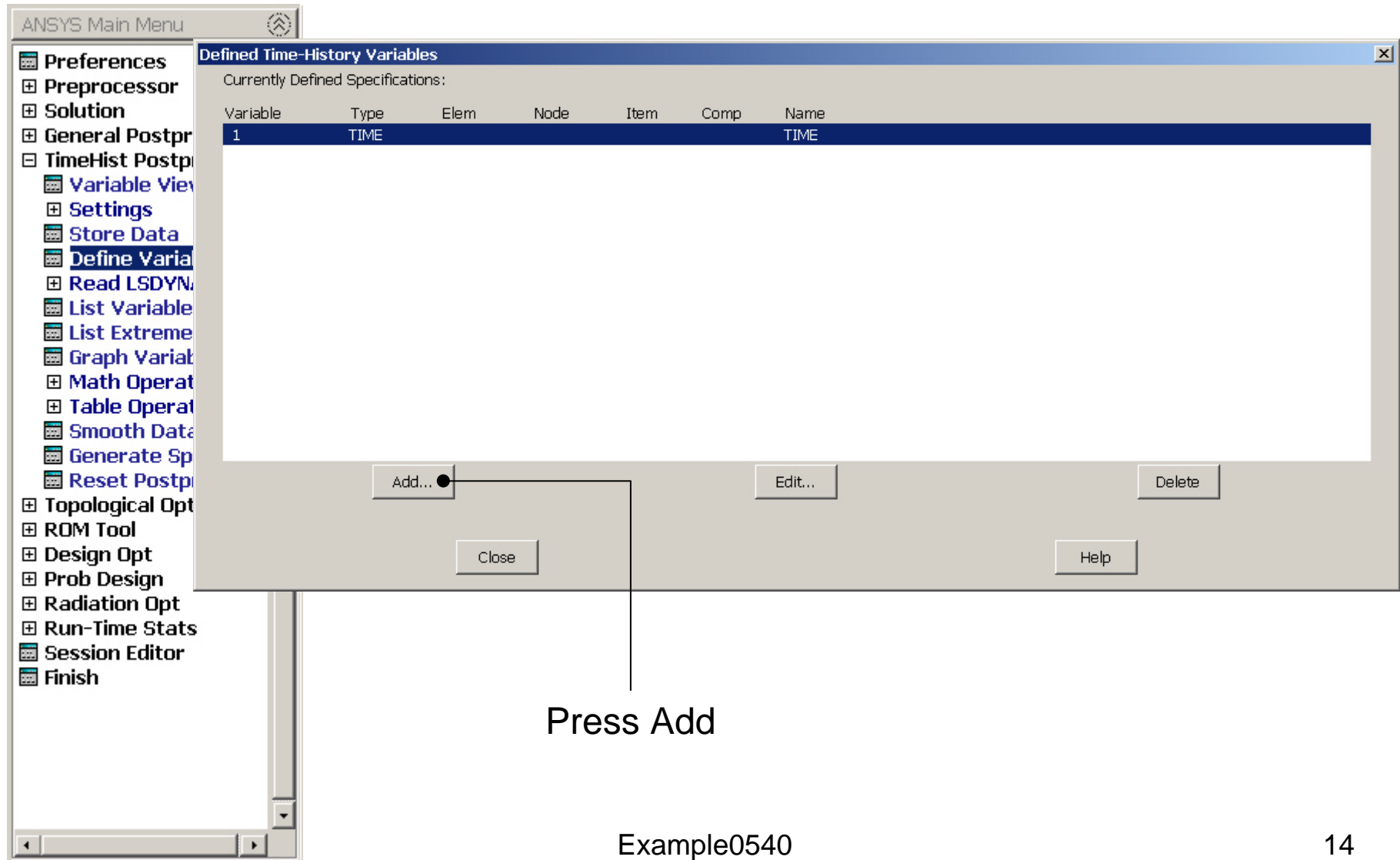
Calculator

Close this dialog box

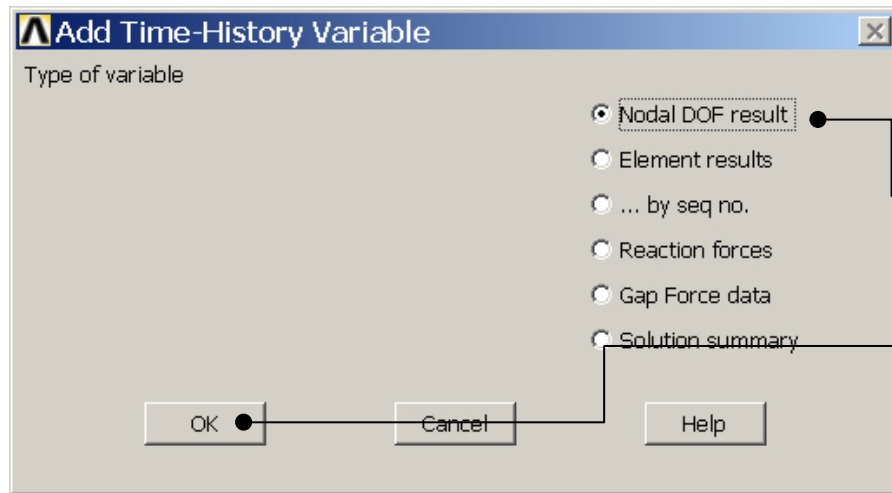
Example0540

13

# Example – Define Variables



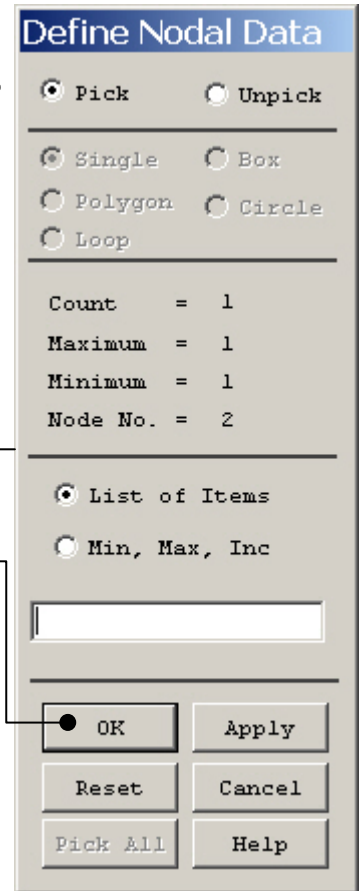
# Example – Add Time-History Var.



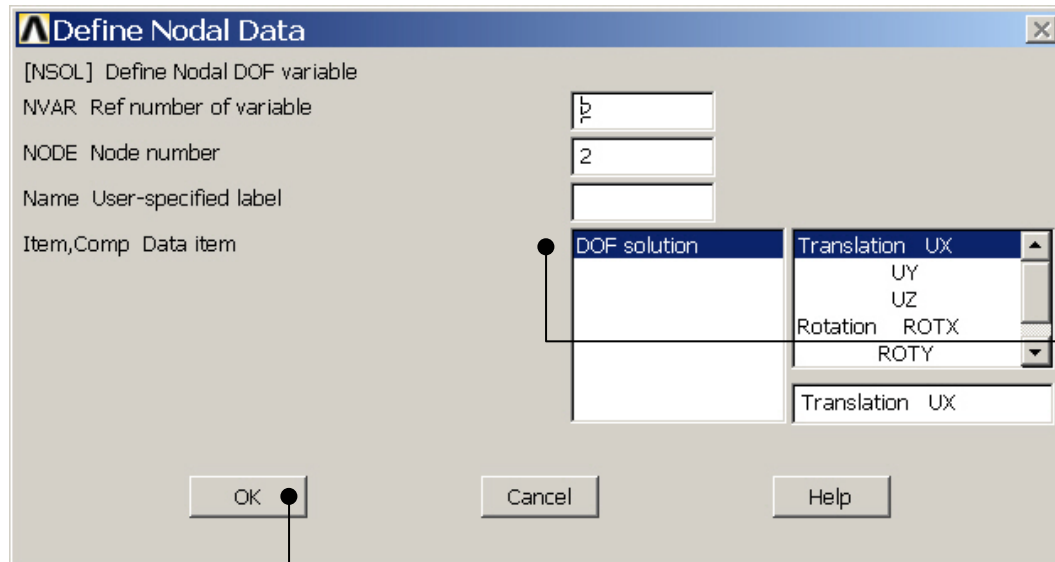
Pick the top right node

Select Nodal DOF result

Press OK

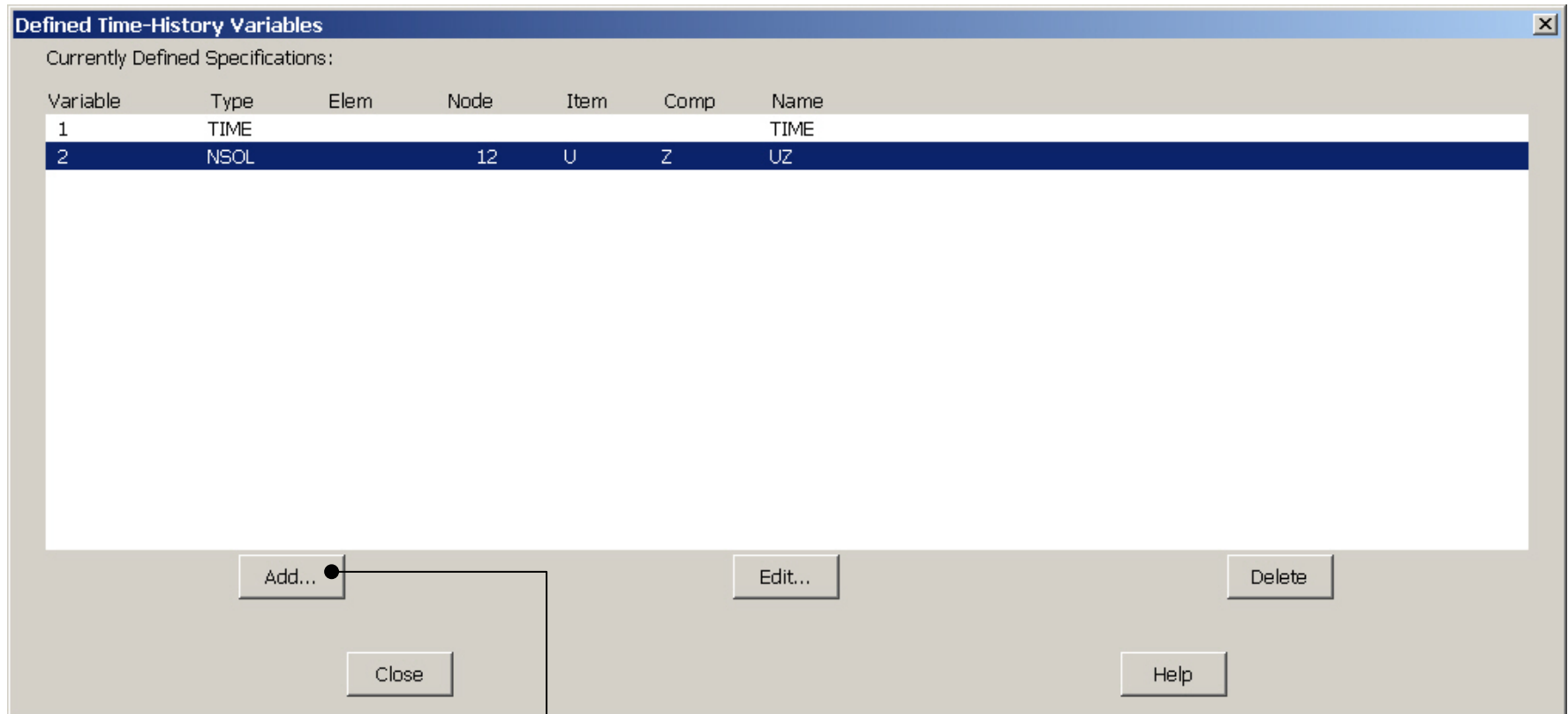


Press OK



Select DOF solution and Translation UZ

# Example – Add Time-History Var.





# Example - Settings

The image shows the ANSYS Main Menu on the left and the Graph Settings dialog box on the right. 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 UZ for the top node on the X-axis (points to the Single variable no. field)
- Press OK (points to the OK button)

Example0540

ANSYS Computational Mechanics, AVO, LSPJerg

# 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, which has opened the 'Axes Modifications for Graph Plots' dialog box. The dialog box contains various settings for graph axes, including labels, ranges, scales, and divisions. The 'OK' button is highlighted, and an arrow points to it with the text 'Press OK'.

**Enter Deformation UZ**

**Enter Force FZ**

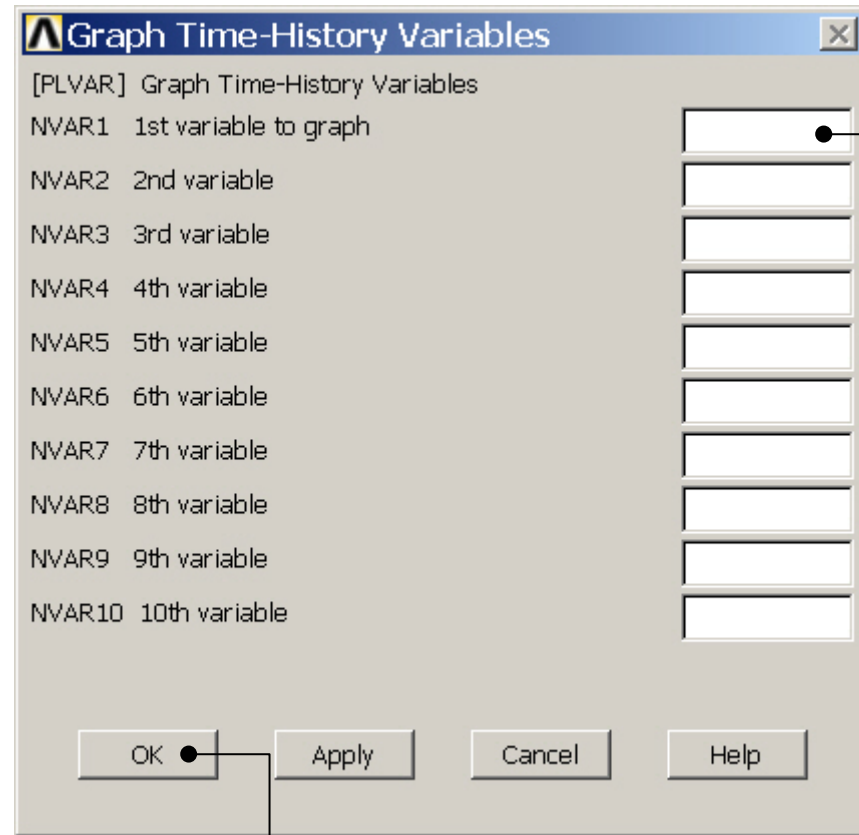
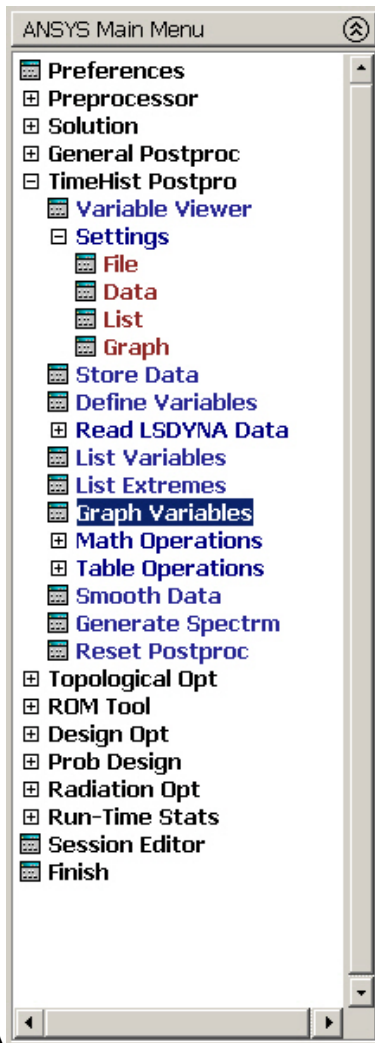
**Press OK**

**ANSYS**

**Example0540**

**18**

# Example – Graph Variables



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

Press OK

Example0540

# Example - Graph

