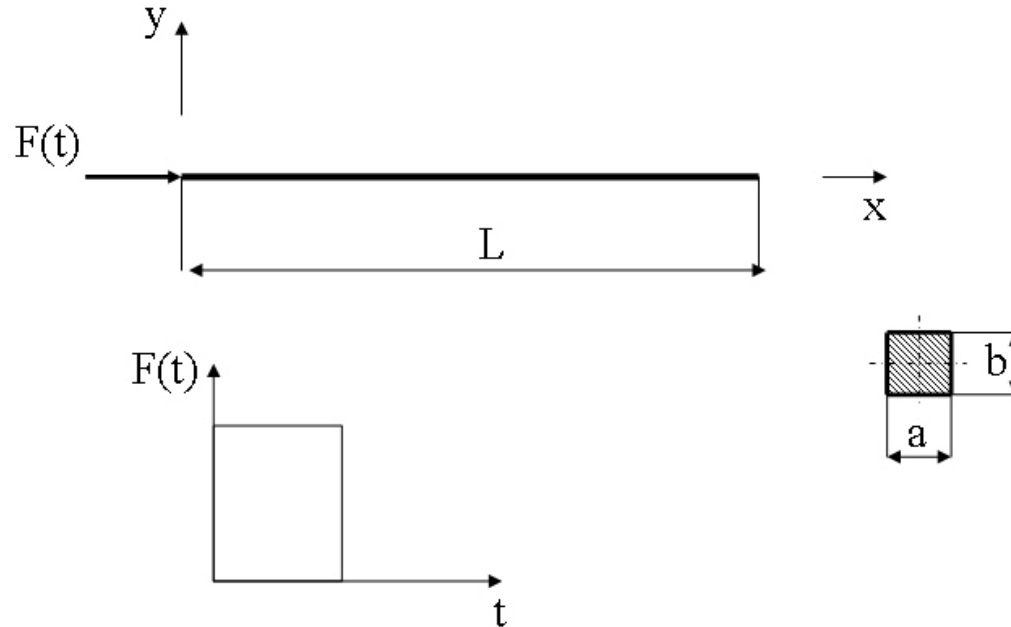


# Course in ANSYS

Example0420

# Example – Rod

**Objective:**

Compute the transient response

**Tasks:**

Perform a transient analysis

**Topics:**

Topics: Start of analysis, Element type, Real constants, Material, modeling, transient analysis

$$E = 200 \text{E}9 \text{N/m}^2$$

$$\nu = 0.3$$

$$L = 1 \text{m}$$

$$I = 0.1^4/12 \text{m}^4$$

$$\rho = 7860 \text{kg/m}^3$$

# Example - title

**Utility Menu > File > Change Jobname**

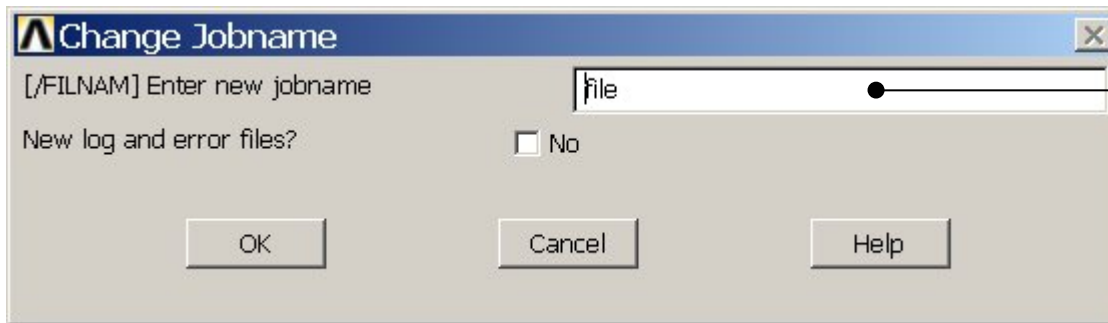


GUI

**/jobname, Example0420**



Command line entry

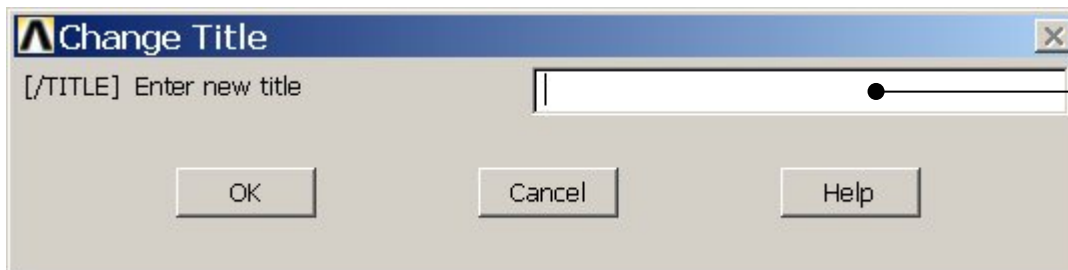


Enter: Example0420

**Utility Menu > File > Change Title**

**/title, Rod**

Enter: Rod



# Example - Keypoints

Note: An empty # result in automatic numbering.

**Preprocessor > Modeling > Create > Keypoints > In Active CS**

/PREP7

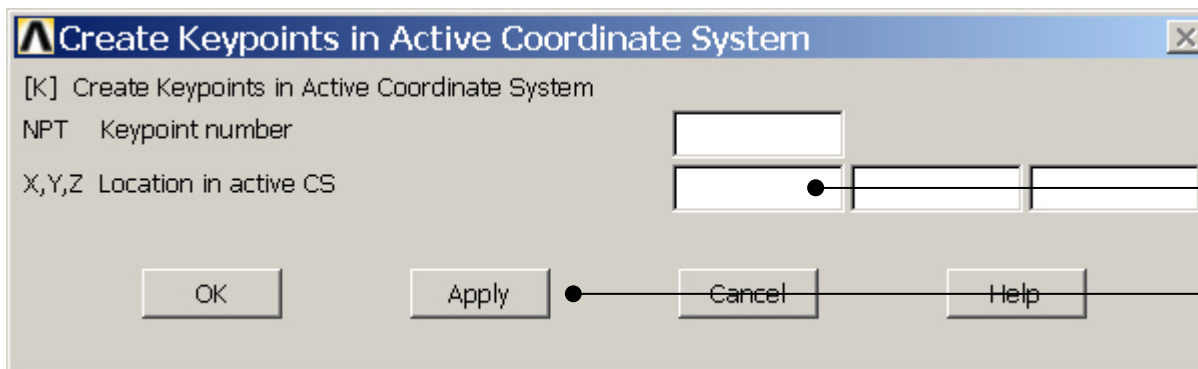
K,,,,

K,,100,,

General format:  
K,#,X,Y,Z

# Keypoint number  
X Keypoint x-coordinate  
Y Keypoint y-coordinate  
Z Keypoint z-coordinate

Enter 0,0,0 and  
Press **Apply**  
Enter 100,0,0 and  
Press **Apply**

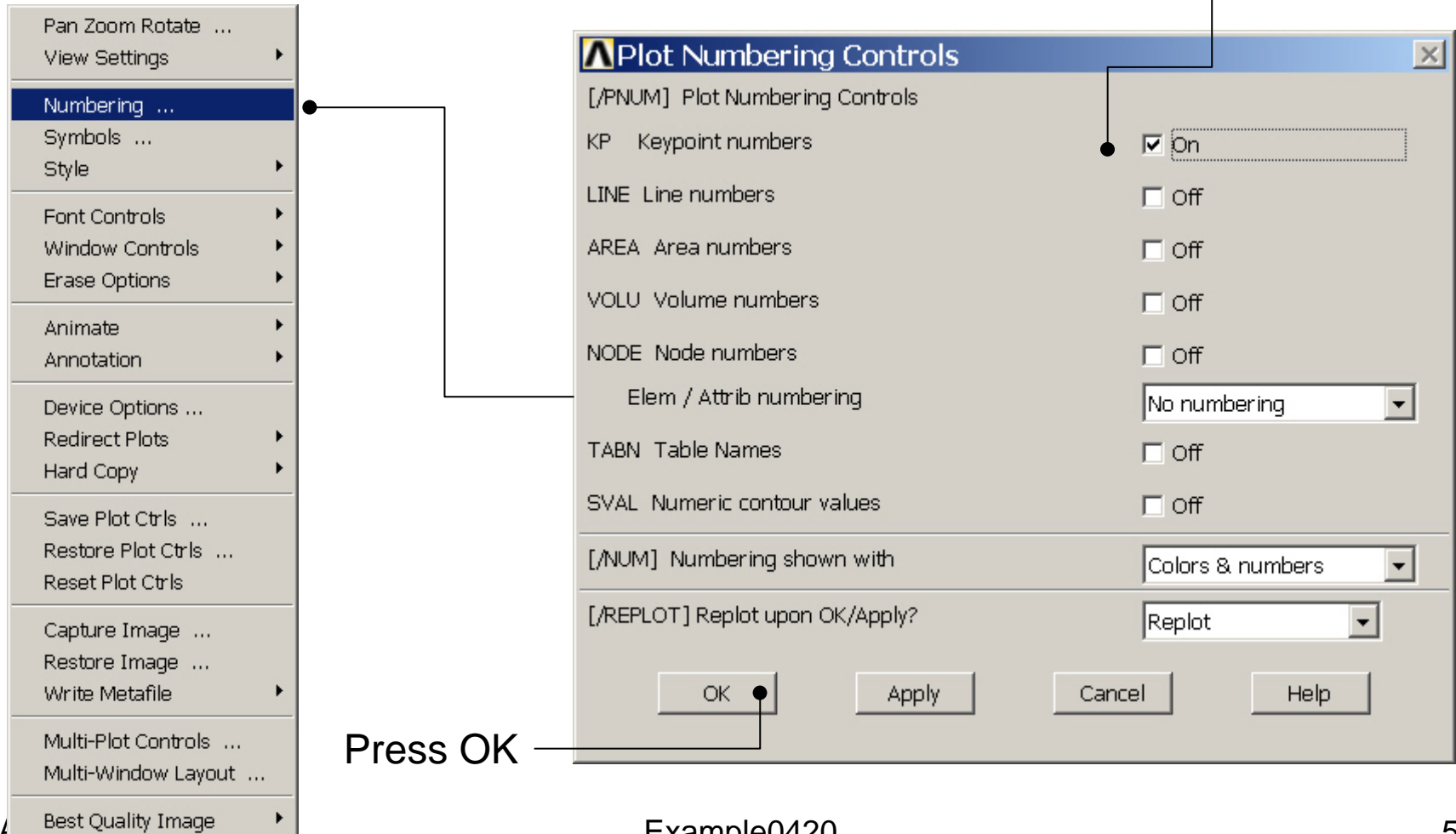


Note: An empty box result in a zero. It is allowed to enter 0.0 in each box.

# Example - Numbering

Utility Menu > PlotCtrls > Numbering

Switch on Keypoint numbers



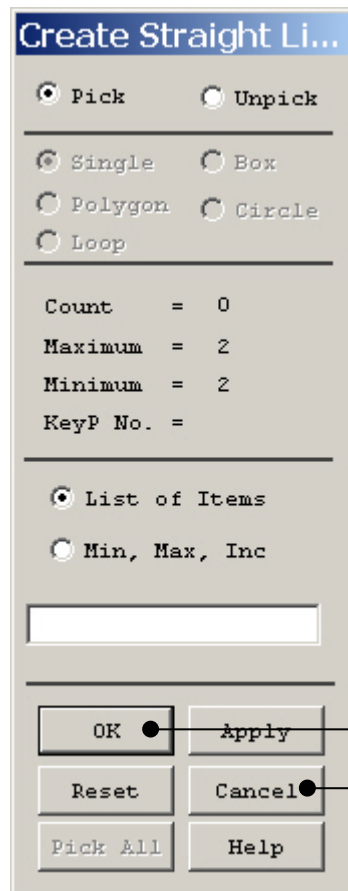
Example0420

# Example - Lines

**Preprocessor > Modeling > Create > Lines > Lines > Straight Line**

Create a line between KP1 and KP2.

L,1,2



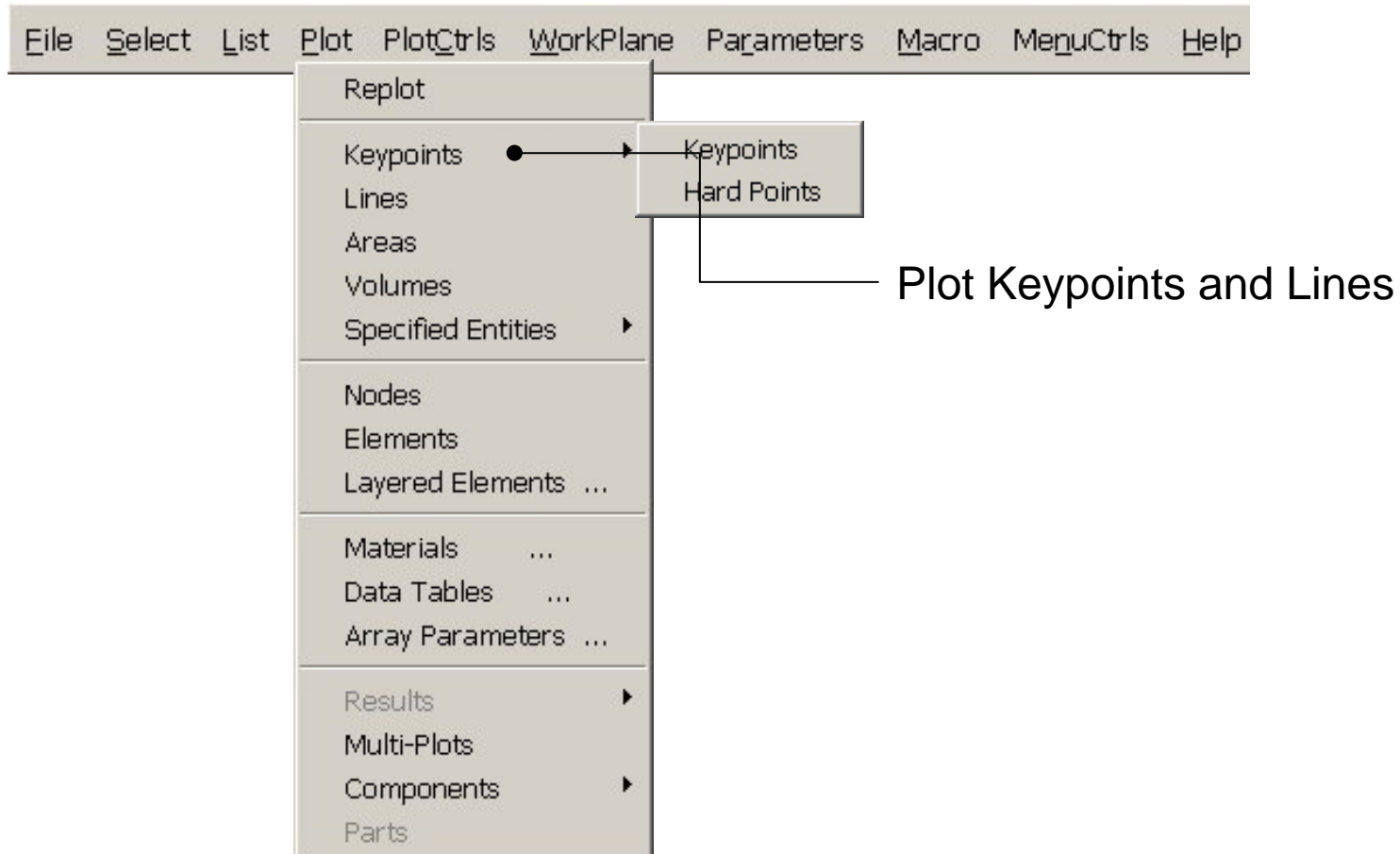
HINT: By clicking with the right-hand mouse button you shift between the Pick/Unpick function. This is indicated by the direction of the cursor arrow:

Pick: upward arrow

Unpick: downward arrow

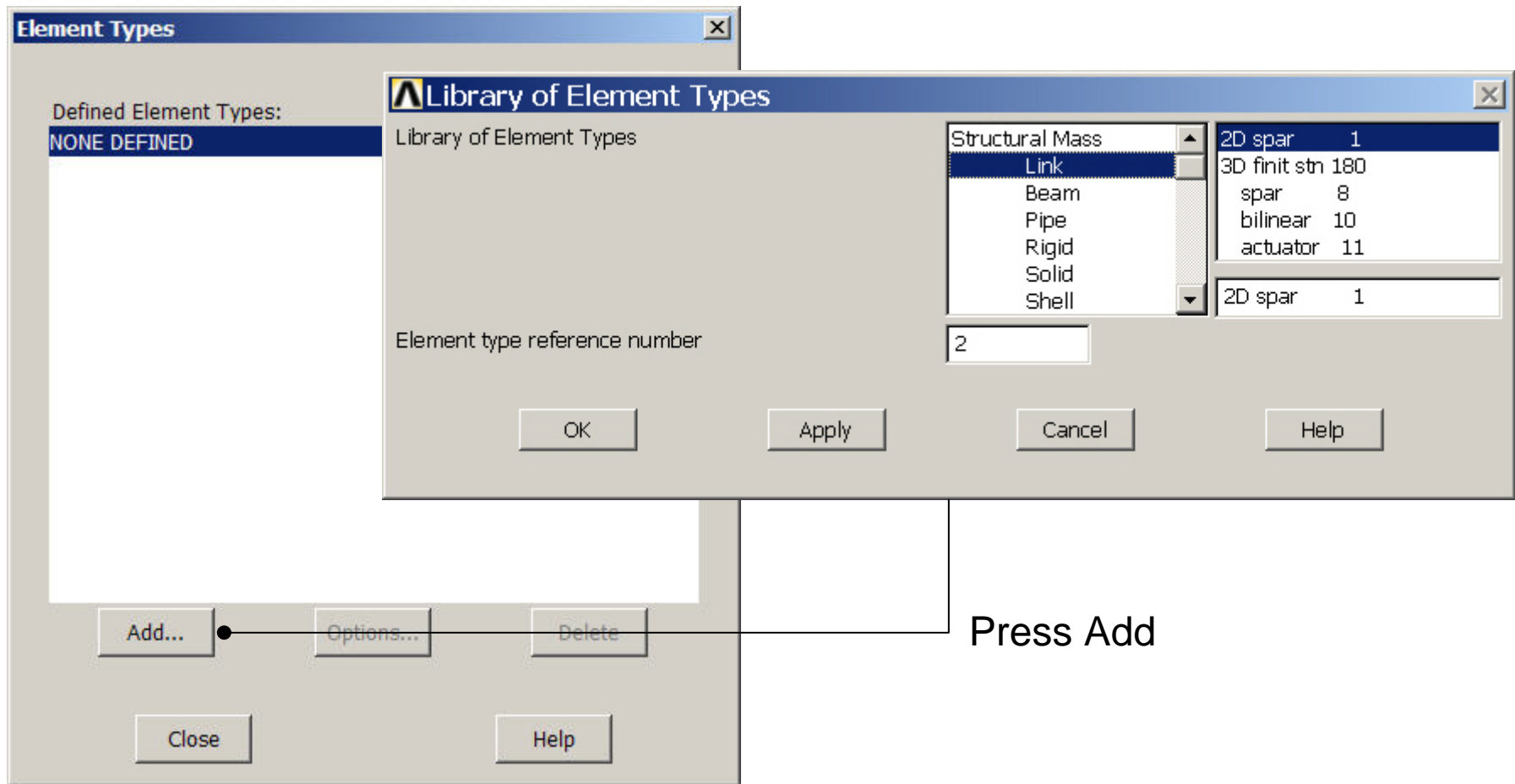
Press OK or Cancel to finish selection

# Example - Plot - Nodes



# Example – Element Type

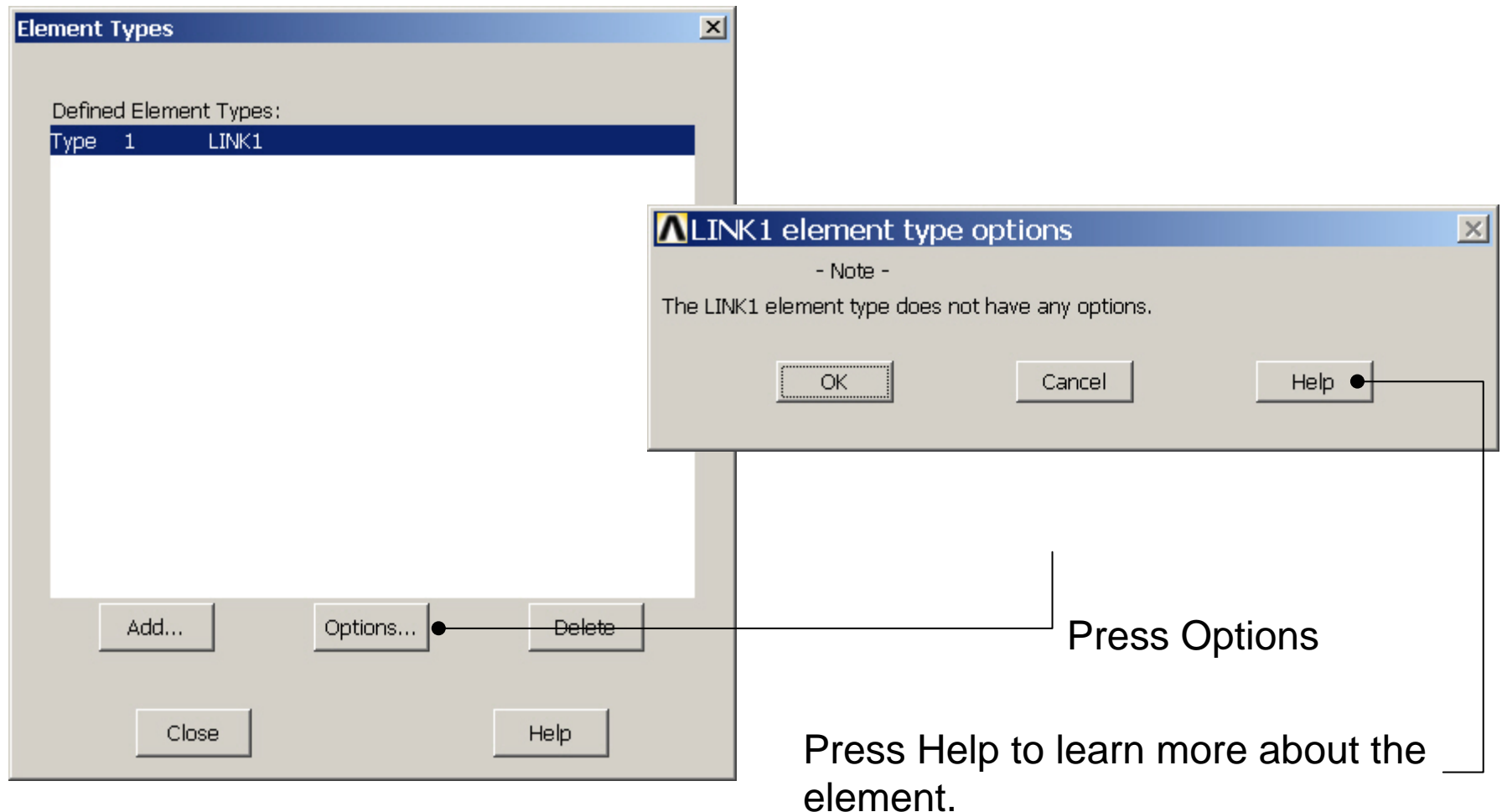
Preprocessor > Element Type > Add/Edit/Delete





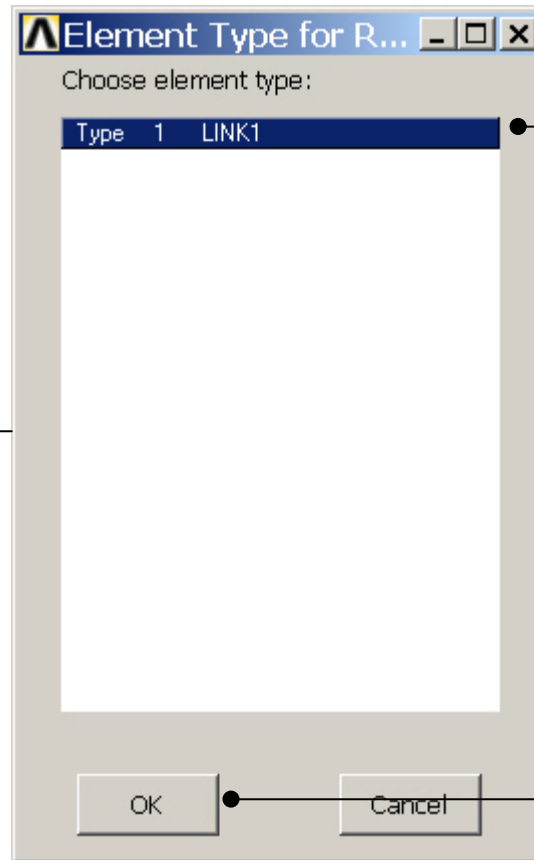
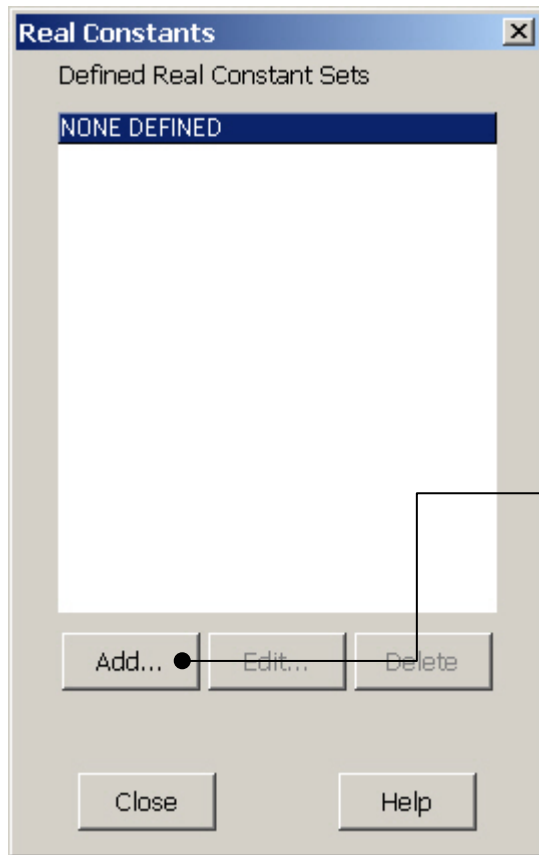
# Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete



# Example – Real Constants

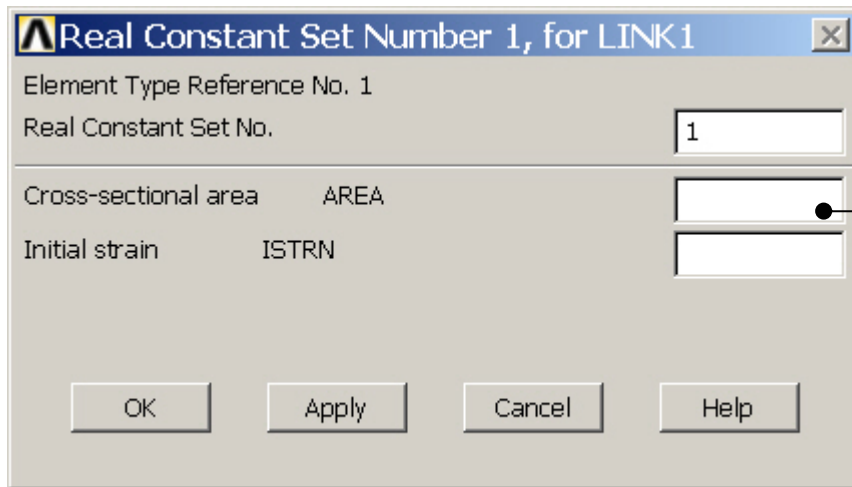
Preprocessor > Real Constants > Add



Place the cursor  
on the relevant  
element and  
press OK

# Example - Real Constants

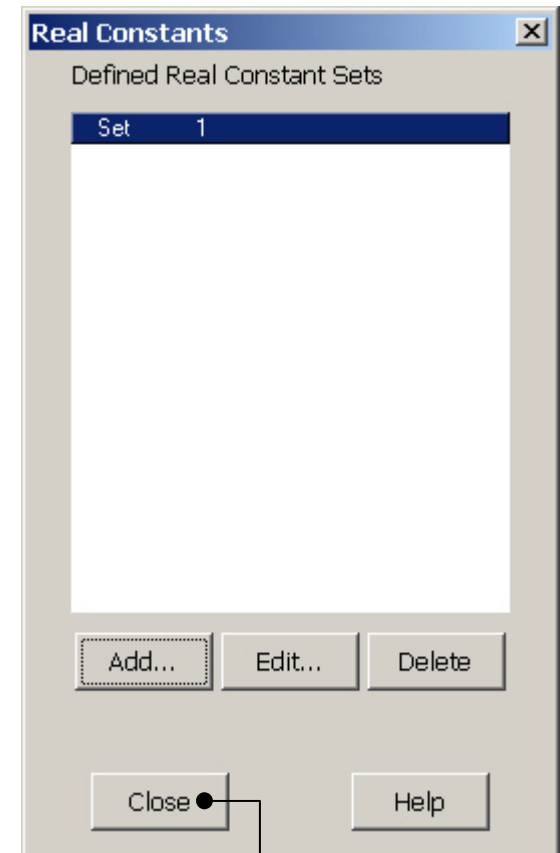
Preprocessor > Real Constants > Add



Enter 1

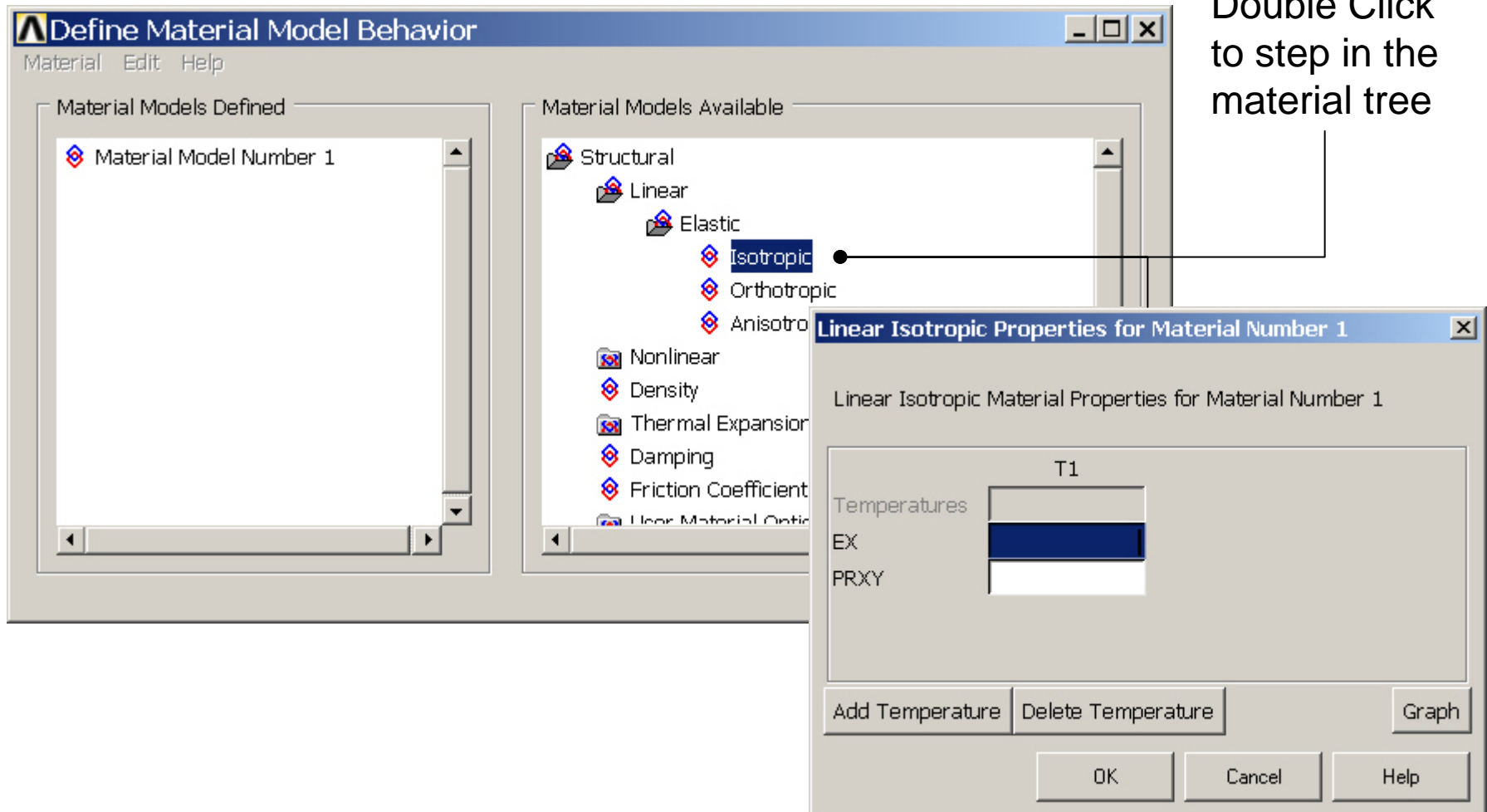
Press OK

Press Close  
to finish



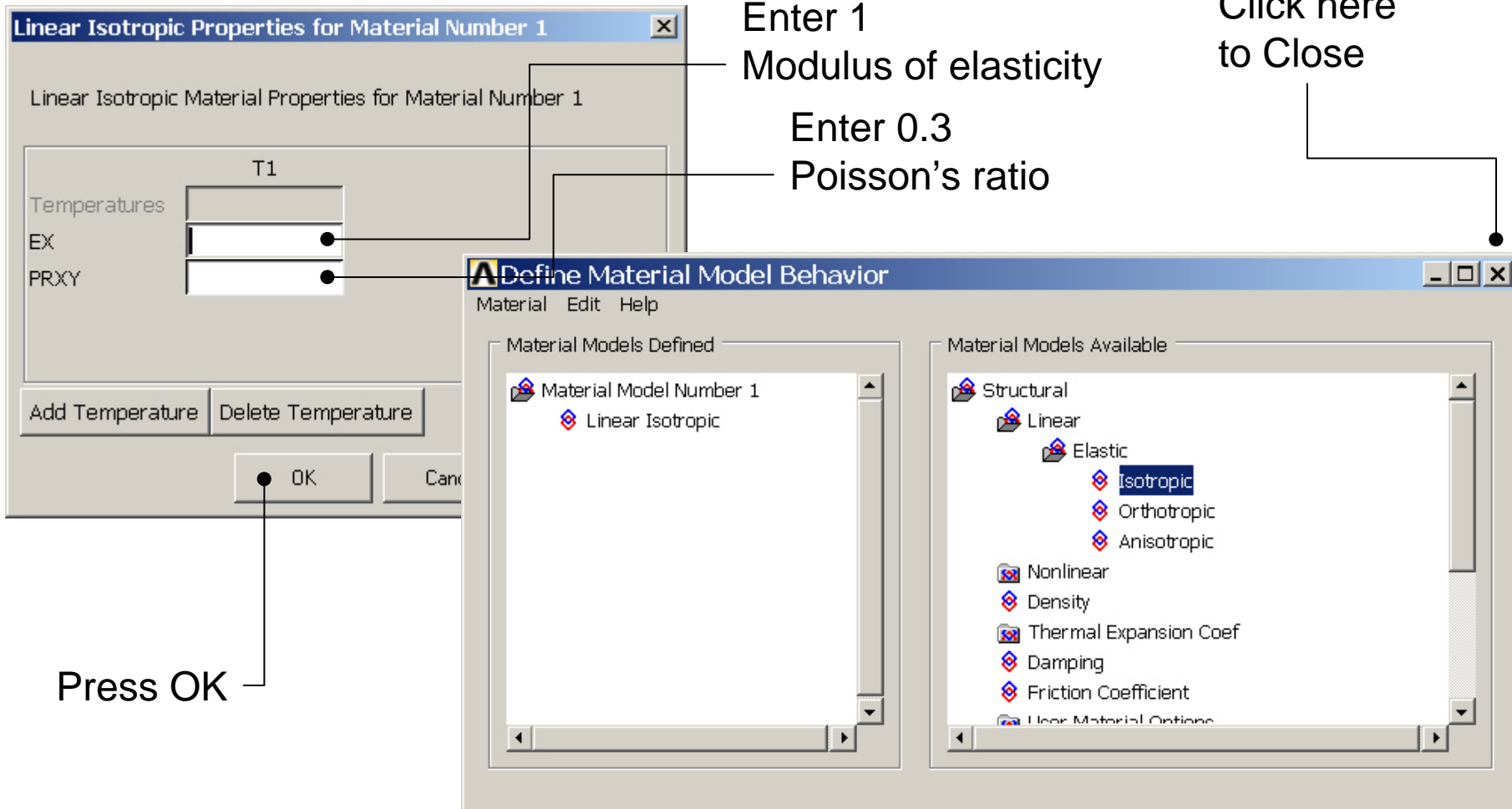
# Example - Material Properties

Preprocessor > Material Props > Material Models

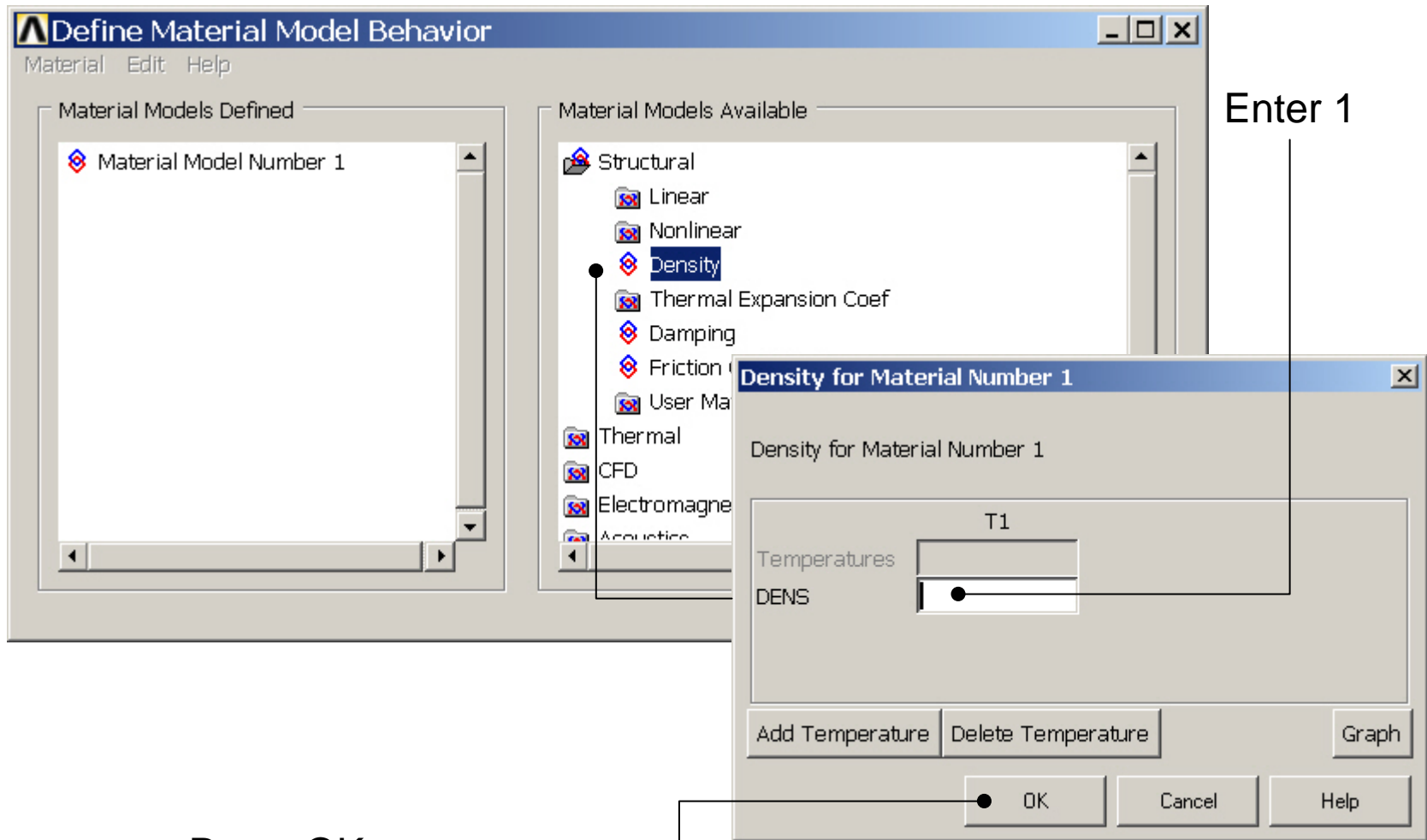


# Example - Material Properties

Preprocessor > Material Props > Material Models



# Example - Density



# Example - Meshing

Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

Select/Pick  
L1

Element Size on P...

☒ Pick ☐ Unpick

☒ Single ☐ Box

☐ Polygon ☐ Circle

☐ Loop

Count = 0

Maximum = 1

Minimum = 1

Line No. =

☒ List of Items

☐ Min, Max, Inc

OK Apply

Reset Cancel

Pick All Help

Element Sizes on Picked Lines

[LESIZE] Element sizes on picked lines

SIZE Element edge length

NDIV No. of element divisions

(NDIV is used only if SIZE is blank or zero)

KYNDIV SIZE,NDIV can be changed ☒ Yes

SPACE Spacing ratio

ANGSIZ Division arc (degrees)

( use ANGSIZ only if number of divisions (NDIV) and element edge length (SIZE) are blank or zero)

Clear attached areas and volumes ☐ No

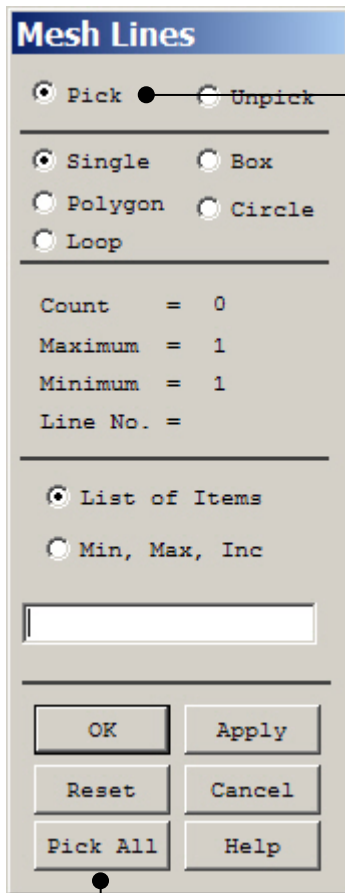
OK Apply Cancel Help

Press OK when finish with selection

Enter 10

# Example - Meshing

Preprocessor > Meshing > Mesh > Lines



Select individual lines to be meshed by Picking

**NB:** It is often necessary to “Clear” the model for example if Element Type is to be changed

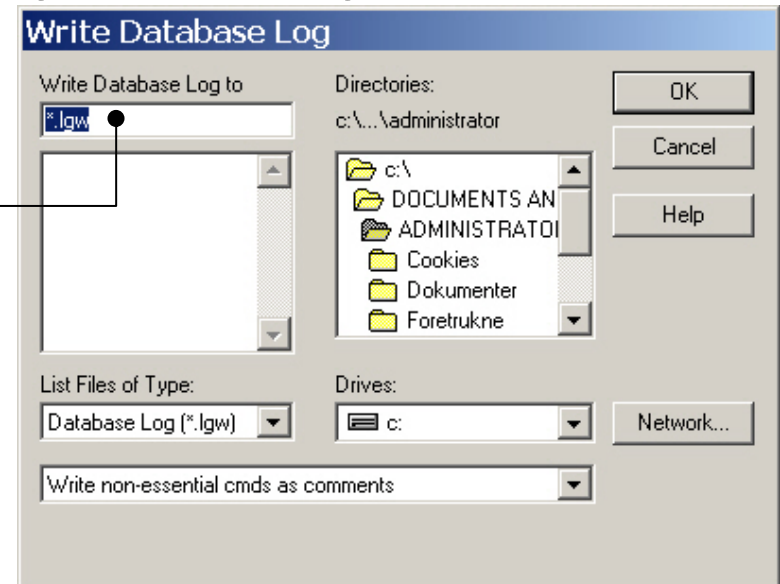
Select all lines defined to be meshed



# Example – Analysis Type

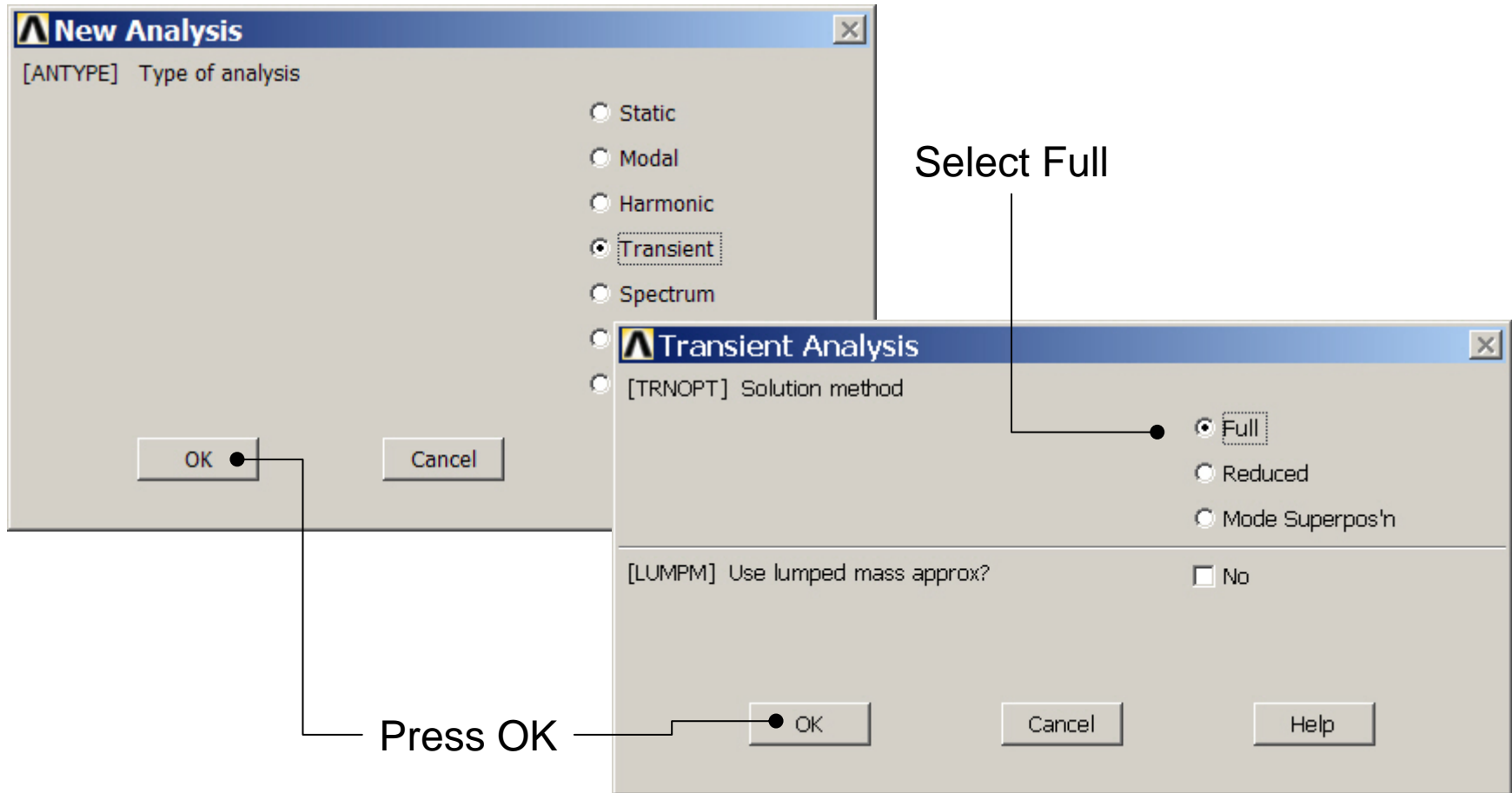
**File > Write DB log file**

Enter “example0420.lgw”

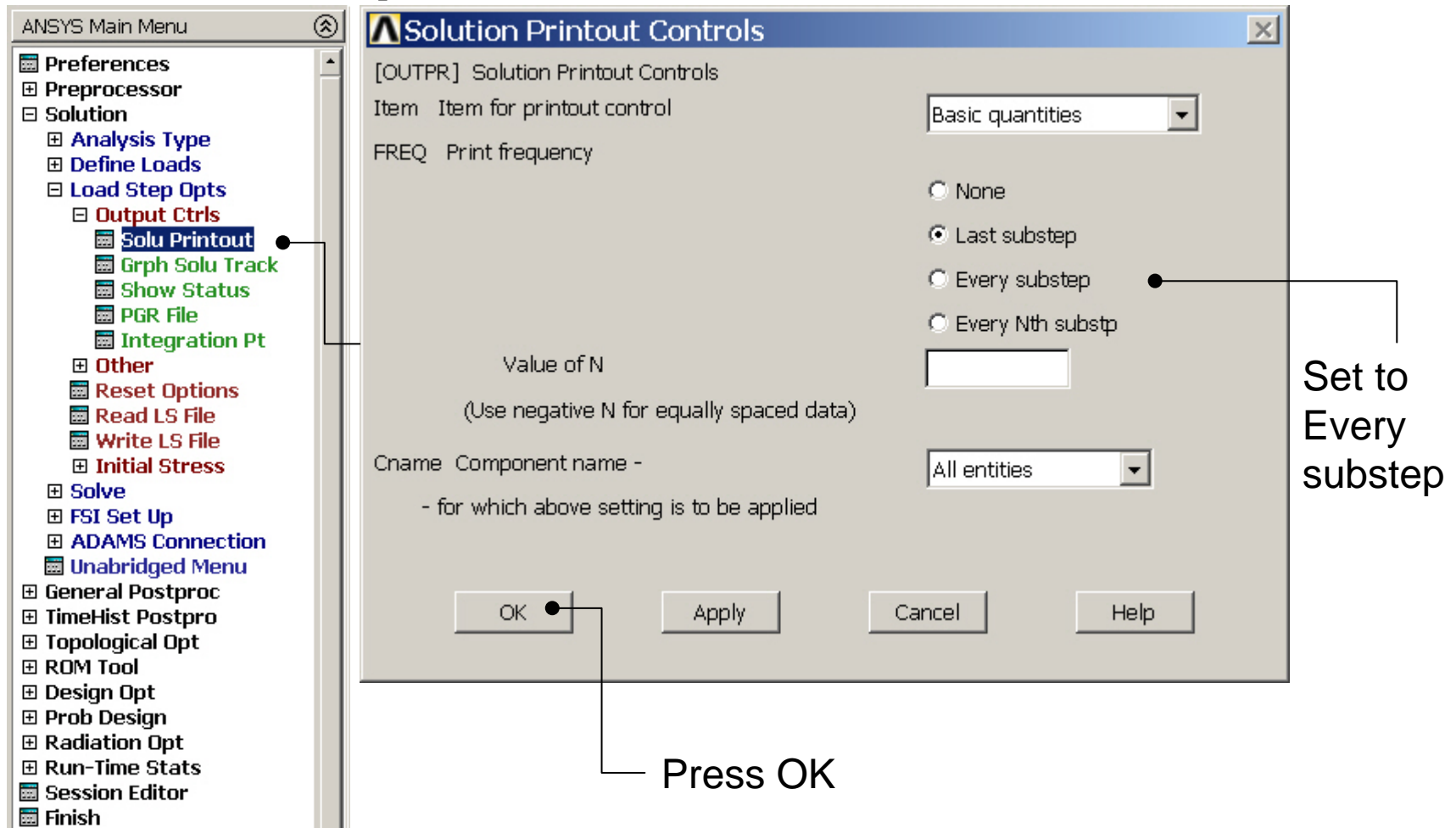


# Example – Analysis Type

**Solution > Analysis Type > New Analysis**



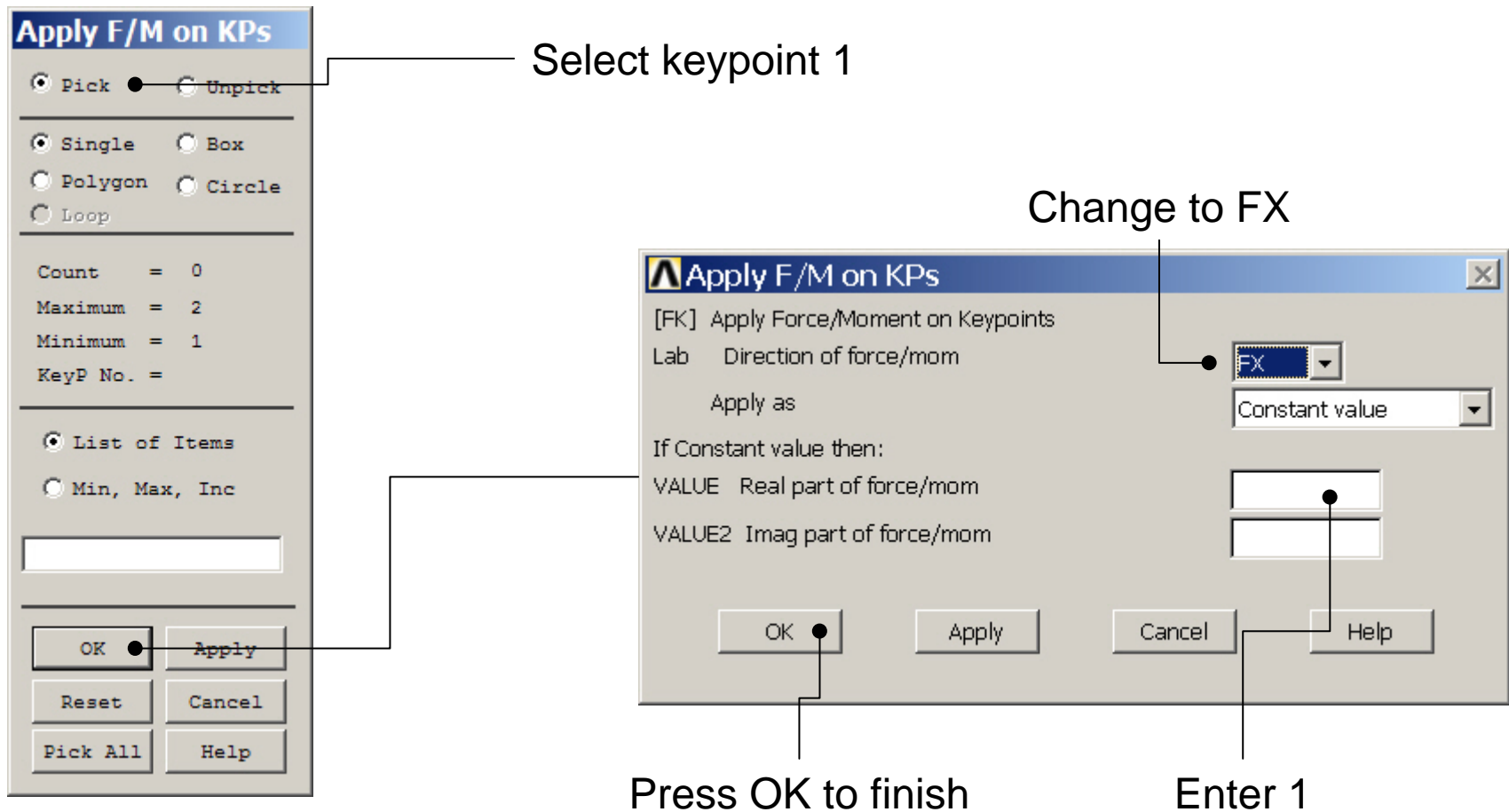
# Example – Printout Controls



Example0420

# Example – Define Loads

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



# Example – Solution Controls

The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Transient' selected in the dropdown and 'Calculate prestress effects' unchecked. The 'Time Control' section has 'Automatic time stepping' set to 'Off', 'Time increment' selected with a value of 0.0001, and 'Time step size' set to 1. The 'Write Items to Results File' section has 'User selected' chosen, with a list containing 'Nodal DOF Solution', 'Nodal Reaction Loads', 'Element Solution', 'Element Nodal Loads', and 'Element Nodal Stresses'. The 'Frequency' section has 'Write every substep' selected, with 'where N = 1' indicated. Annotations with arrows point to these settings: 'Enter 0.0001' points to the 'Time increment' field; 'Set Automatic time stepping Off' points to the 'Automatic time stepping' dropdown; 'Set to Time increment' points to the 'Time increment' field; 'Check if set to Write every substep' points to the 'Write every substep' dropdown; and 'Enter 0.0001' points to the 'where N = 1' field.

**Solution Controls**

Basic | Transient | Sol'n Options | Nonlinear | Advanced NL

**Analysis Options**

Small Displacement Transient

☐ Calculate prestress effects

**Time Control**

Time at end of loadstep

Automatic time stepping: Off

☐ Number of substeps

☒ Time increment: 0.0001

Time step size: 1

Minimum time step: 0

Maximum time step: 0

**Write Items to Results File**

☐ All solution items

☐ Basic quantities

☒ User selected

Nodal DOF Solution

Nodal Reaction Loads

Element Solution

Element Nodal Loads

Element Nodal Stresses

Frequency:

Write every substep

where N = 1

OK Cancel Help

Enter 0.0001

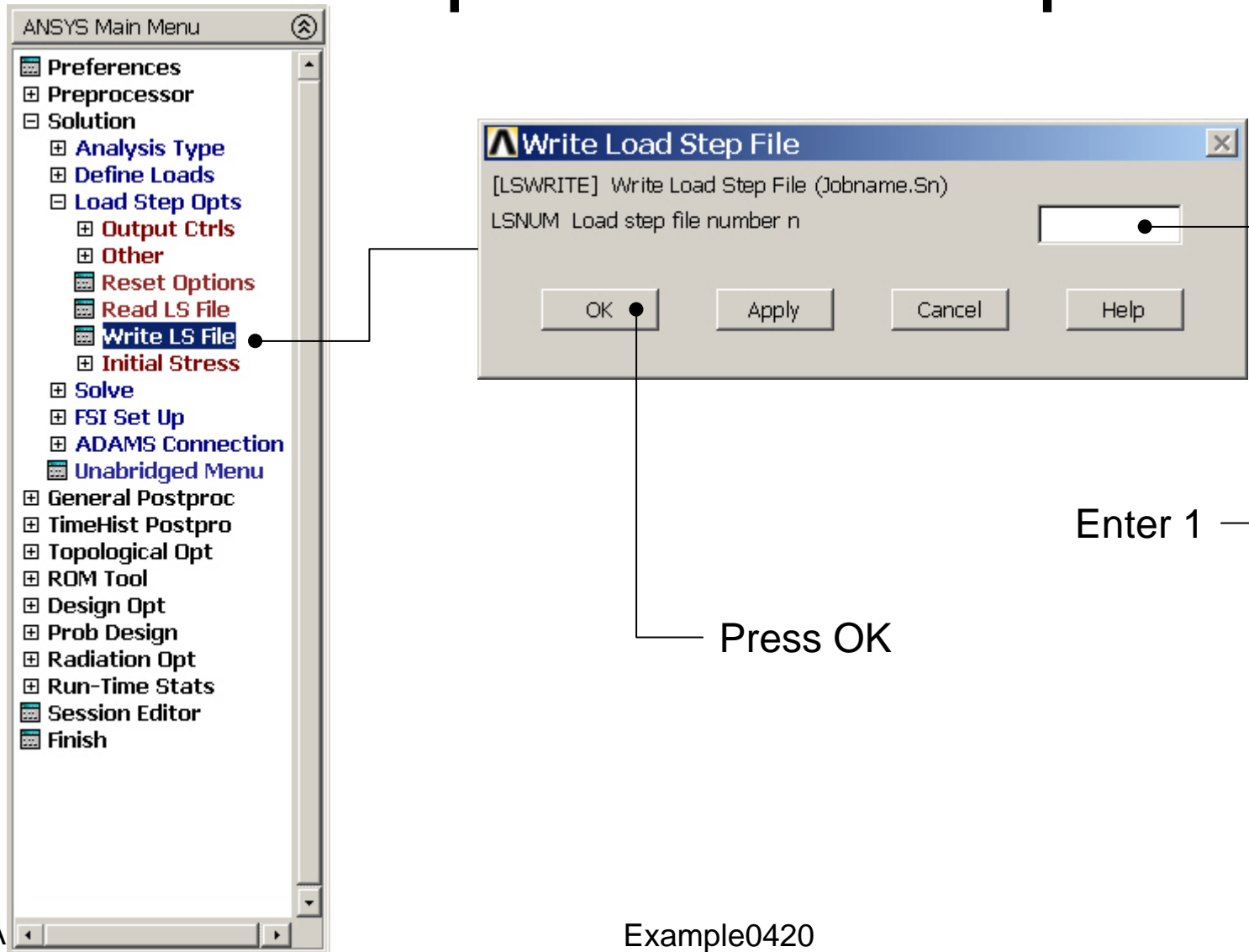
Set Automatic time stepping Off

Set to Time increment

Check if set to Write every substep

Enter 0.0001

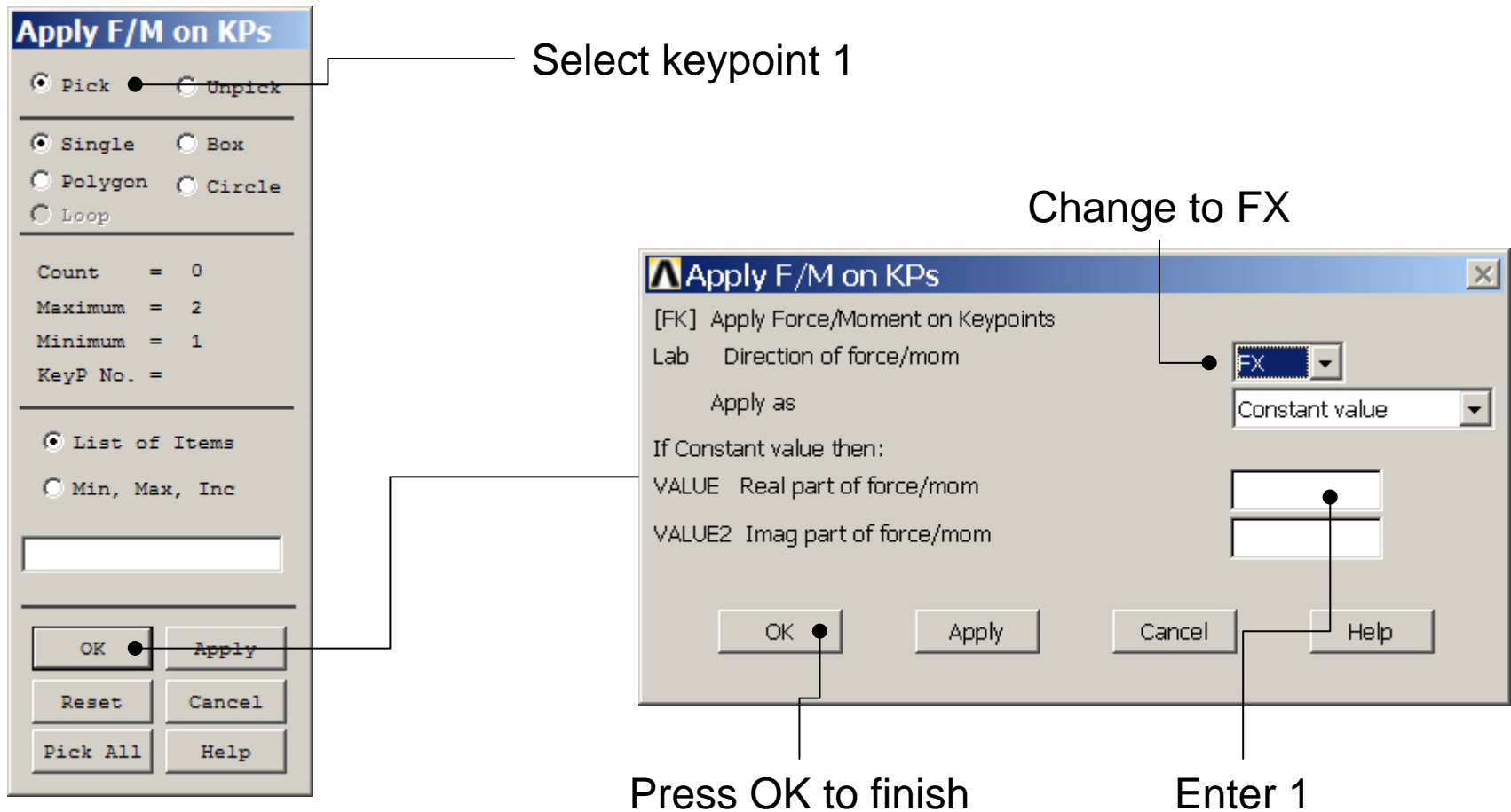
# Example – Loadstep file



Example0420

# Example – Define Loads

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



# Example – Solution Controls

The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Transient' selected in the dropdown and 'Calculate prestress effects' unchecked. The 'Time Control' section has 'Automatic time stepping' set to 'Off', 'Time increment' selected with a value of 30, and 'Time step size' set to 1. The 'Write Items to Results File' section has 'User selected' chosen, with a list containing 'Nodal DOF Solution', 'Nodal Reaction Loads', 'Element Solution', 'Element Nodal Loads', and 'Element Nodal Stresses'. The 'Frequency' section has 'Write every substep' selected and 'where N = 1'. Annotations on the right side point to these settings: 'Enter 30' points to the 'Time increment' value; 'Set Automatic time stepping Off' points to the 'Automatic time stepping' dropdown; 'Set to Time increment' points to the 'Time increment' value; 'Check if set to Write every substep' points to the 'Write every substep' dropdown; and 'Enter 1' points to the 'where N = 1' value.

**Solution Controls**

Basic | Transient | Sol'n Options | Nonlinear | Advanced NL

**Analysis Options**

Small Displacement Transient

☐ Calculate prestress effects

**Time Control**

Time at end of loadstep

Automatic time stepping: Off

☐ Number of substeps

☒ Time increment: 30

Time step size: 1

Minimum time step: 0

Maximum time step: 0

**Write Items to Results File**

☐ All solution items

☐ Basic quantities

☒ User selected

Nodal DOF Solution

Nodal Reaction Loads

Element Solution

Element Nodal Loads

Element Nodal Stresses

Frequency:

Write every substep

where N = 1

OK Cancel Help

Enter 30

Set Automatic time stepping Off

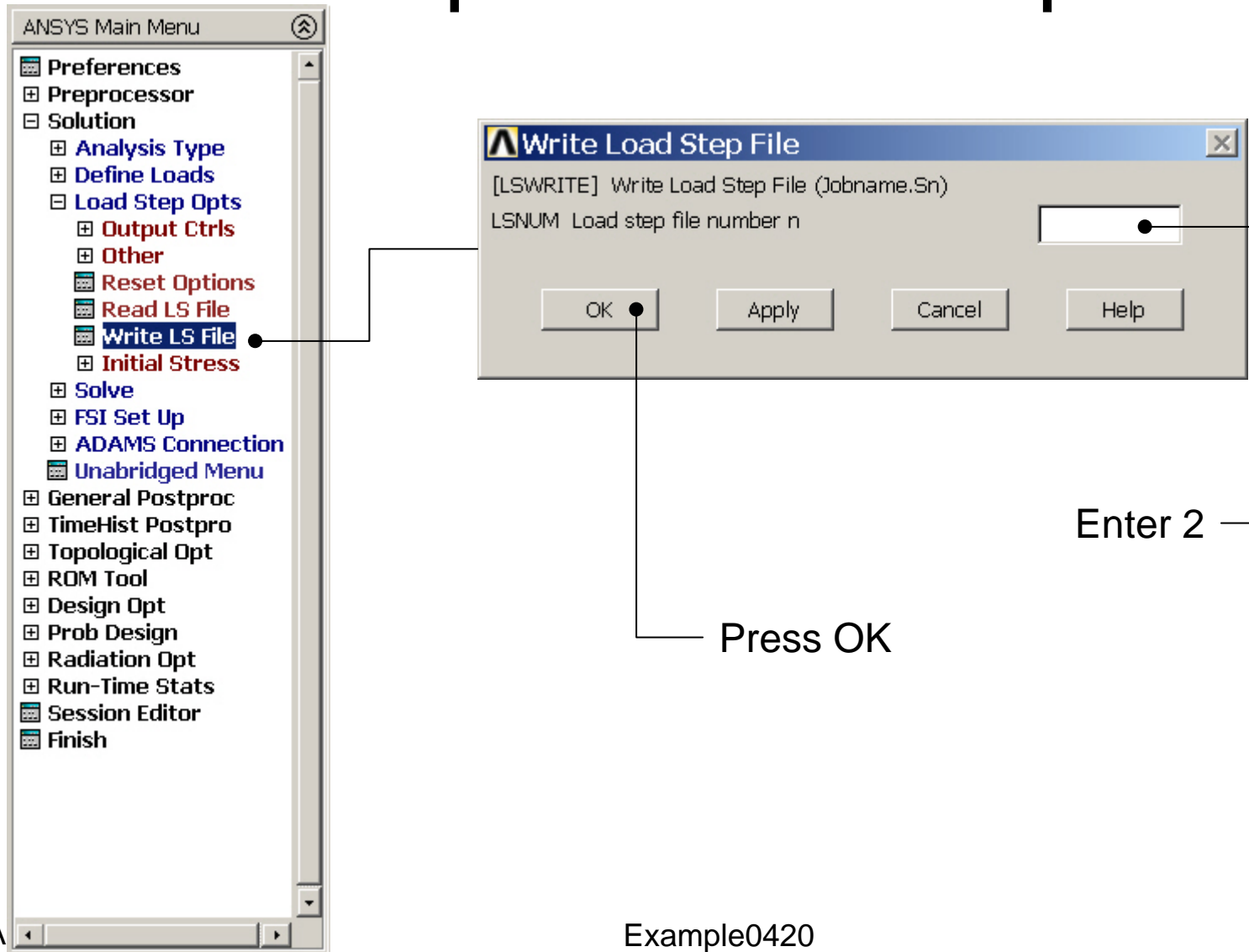
Set to Time increment

Check if set to Write every substep

Enter 1



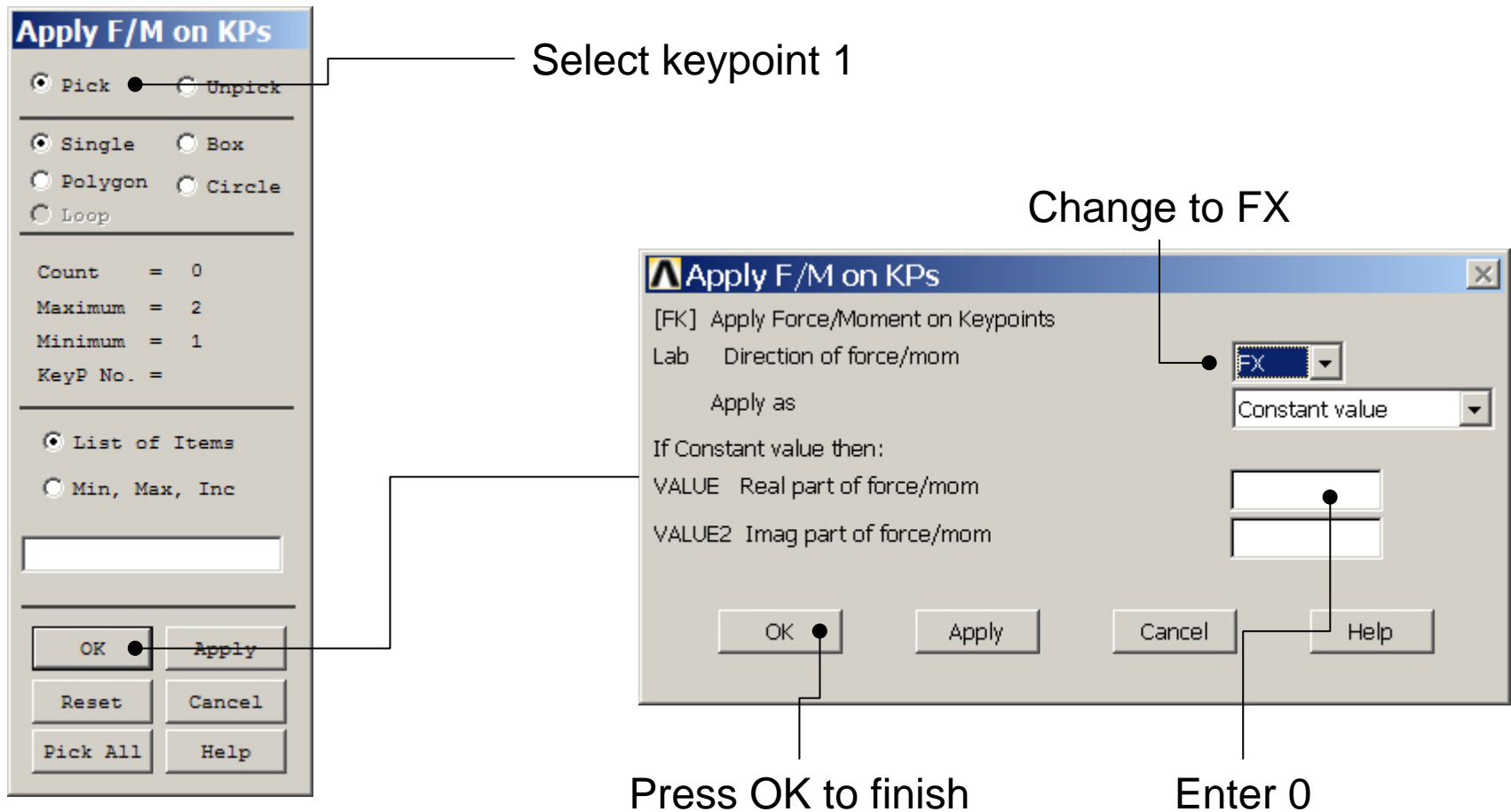
# Example – Loadstep file



Example0420

# Example – Define Loads

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



# Example – Solution Controls

The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Transient' selected in the dropdown and 'Calculate prestress effects' unchecked. The 'Time Control' section has 'Automatic time stepping' set to 'Off', 'Time increment' selected with a value of 30.001, and 'Time step size' set to 1. The 'Write Items to Results File' section has 'User selected' chosen, with a list containing 'Nodal DOF Solution', 'Nodal Reaction Loads', 'Element Solution', 'Element Nodal Loads', and 'Element Nodal Stresses'. The 'Frequency' section has 'Write every substep' selected, with 'where N = 1' indicated. Annotations on the right side point to these specific settings: 'Enter 30.001' points to the 'Time increment' value; 'Set Automatic time stepping Off' points to the 'Automatic time stepping' dropdown; 'Set to Time increment' points to the 'Time increment' value; 'Check if set to Write every substep' points to the 'Write every substep' dropdown; and 'Enter 0.001' points to the 'Time step size' value.

Enter 30.001

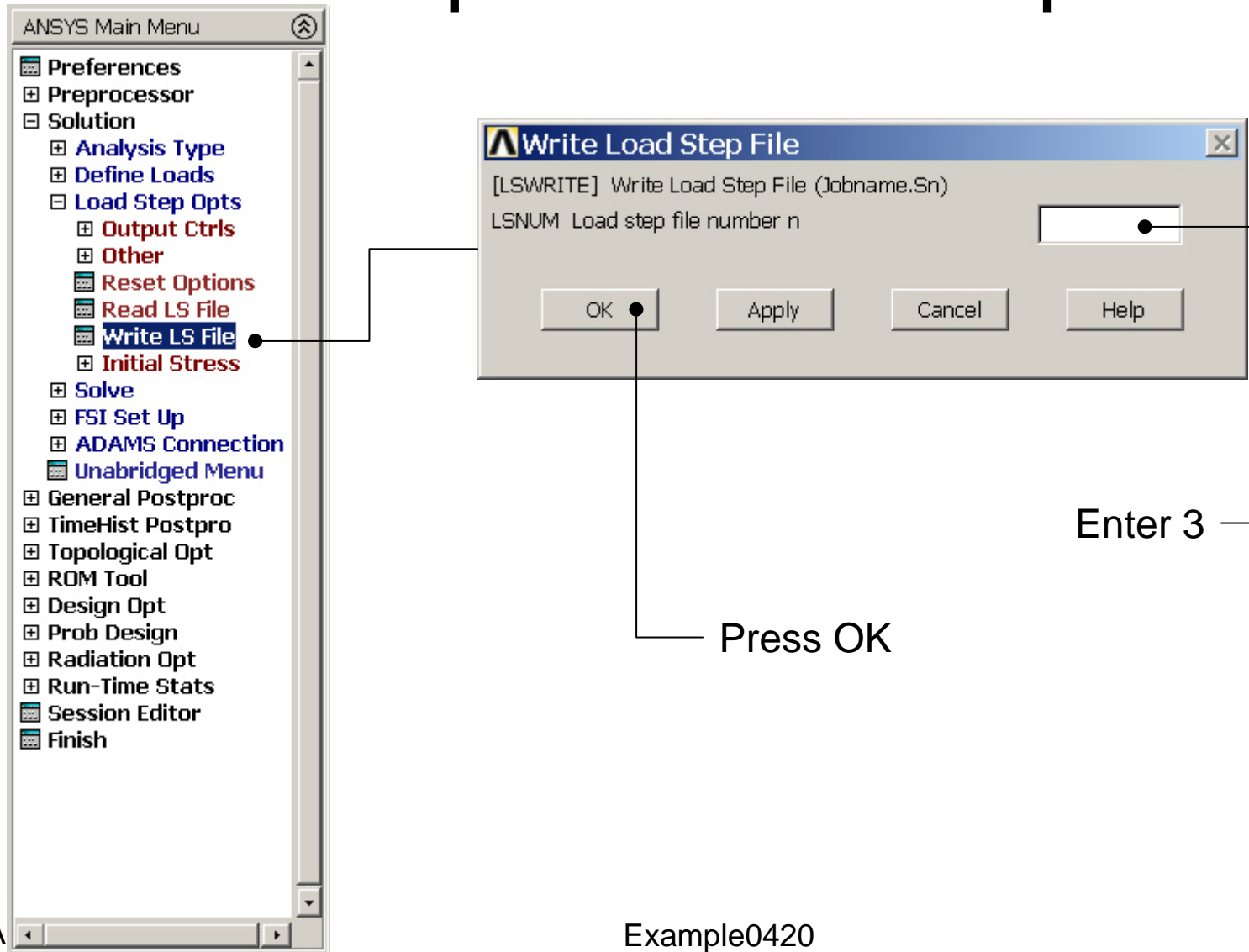
Set Automatic time stepping Off

Set to Time increment

Check if set to Write every substep

Enter 0.001

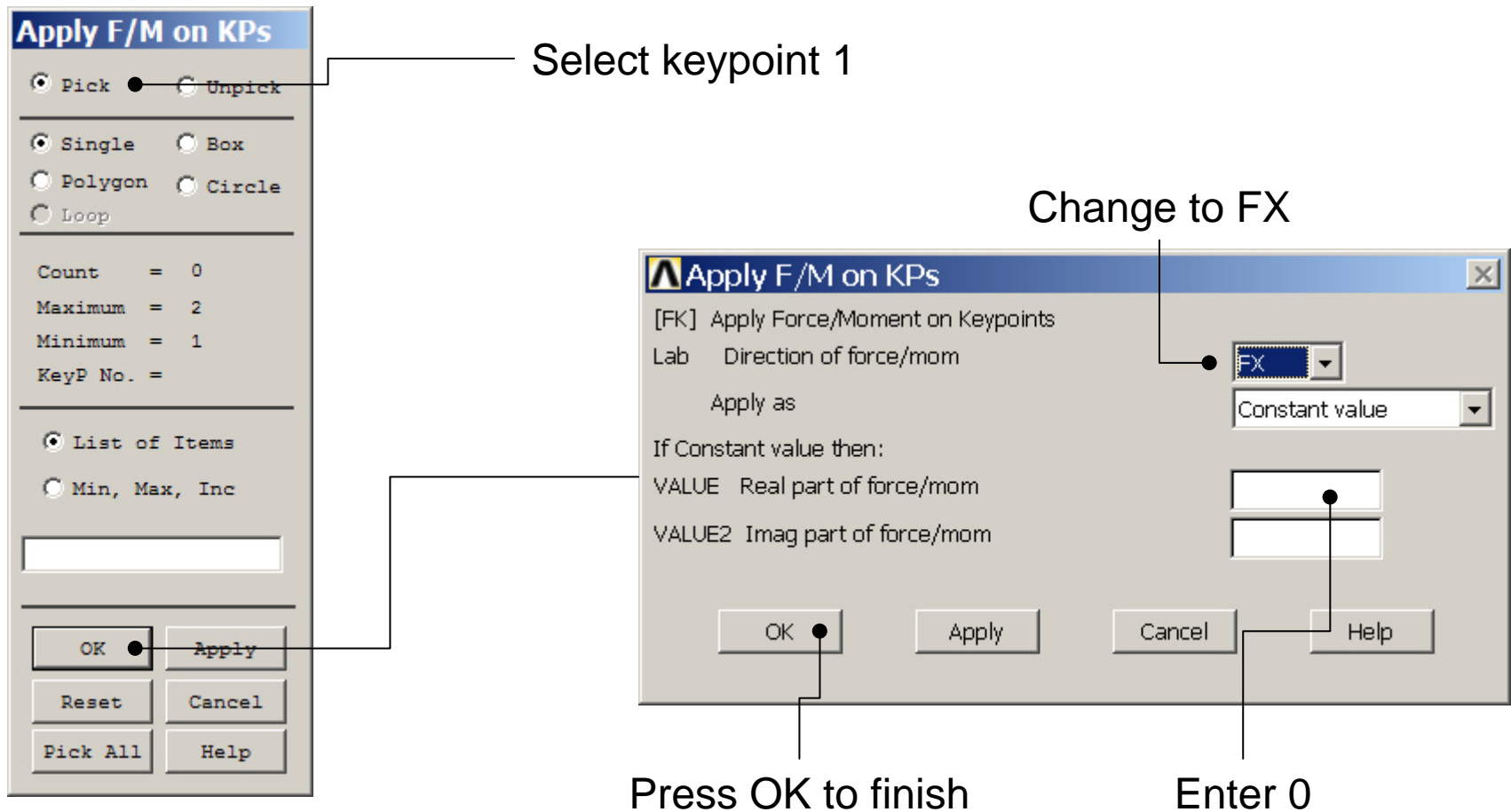
# Example – Loadstep file



Example0420

# Example – Define Loads

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



# Example – Solution Controls

The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Transient' selected in the dropdown and 'Calculate prestress effects' unchecked. The 'Time Control' section has 'Automatic time stepping' set to 'Off', 'Time increment' selected with a value of 250, and 'Time step size' set to 1. The 'Write Items to Results File' section has 'User selected' chosen, with a list containing 'Nodal DOF Solution', 'Nodal Reaction Loads', 'Element Solution', 'Element Nodal Loads', and 'Element Nodal Stresses'. The 'Frequency' section has 'Write every substep' selected and 'where N = 1'. Annotations with leader lines point to these specific settings: 'Enter 250' points to the 'Time increment' value; 'Set Automatic time stepping Off' points to the 'Automatic time stepping' dropdown; 'Set to Time increment' points to the 'Time increment' value; 'Check if set to Write every substep' points to the 'Write every substep' dropdown; and 'Enter 1' points to the 'where N = 1' value.

**Solution Controls**

Basic | Transient | Sol'n Options | Nonlinear | Advanced NL

**Analysis Options**

Small Displacement Transient

☐ Calculate prestress effects

**Time Control**

Time at end of loadstep

Automatic time stepping: Off

☐ Number of substeps

☒ Time increment: 250

Time step size: 1

Minimum time step: 0

Maximum time step: 0

**Write Items to Results File**

☐ All solution items

☐ Basic quantities

☒ User selected

Nodal DOF Solution

Nodal Reaction Loads

Element Solution

Element Nodal Loads

Element Nodal Stresses

Frequency:

Write every substep

where N = 1

OK Cancel Help

Enter 250

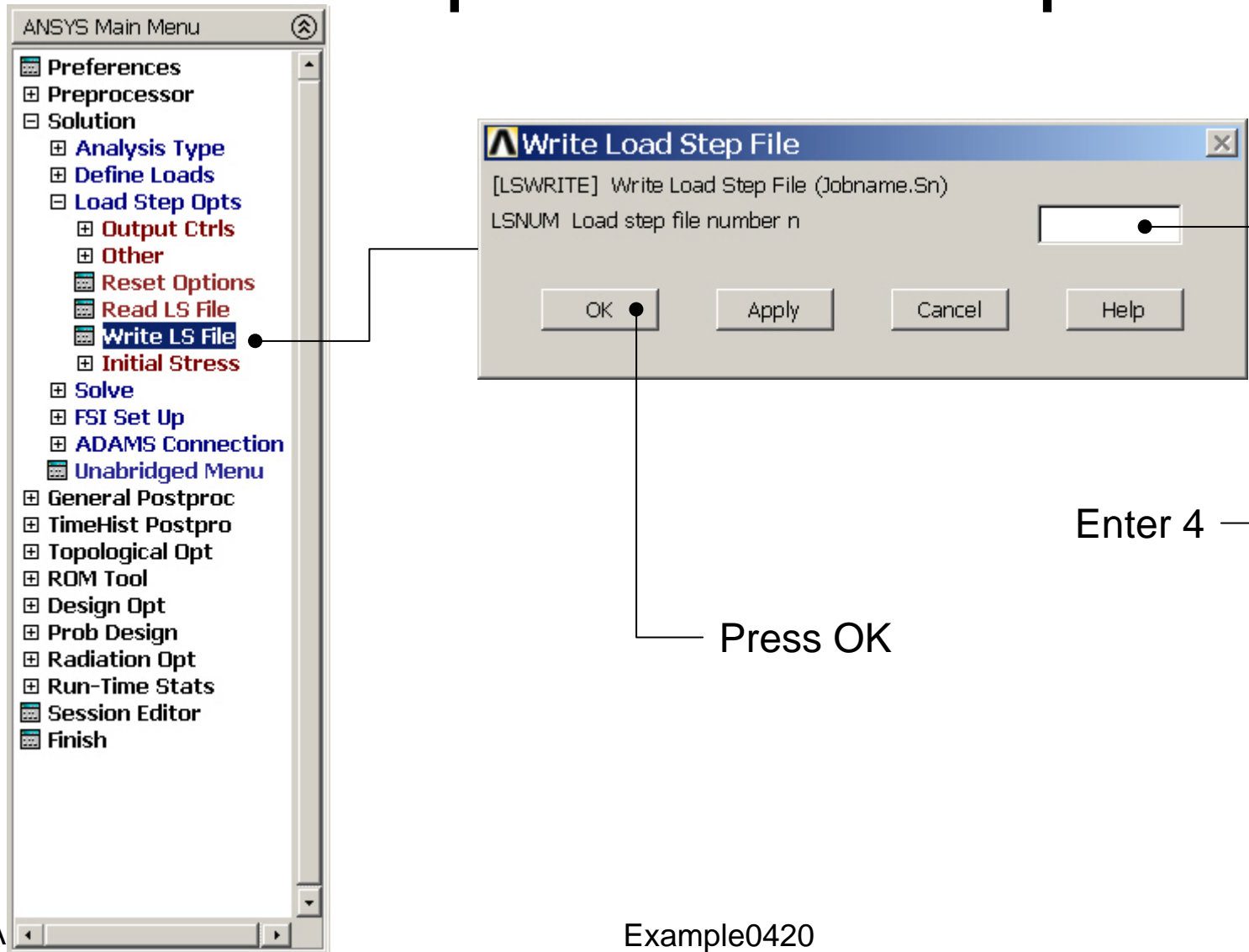
Set Automatic time stepping Off

Set to Time increment

Check if set to Write every substep

Enter 1

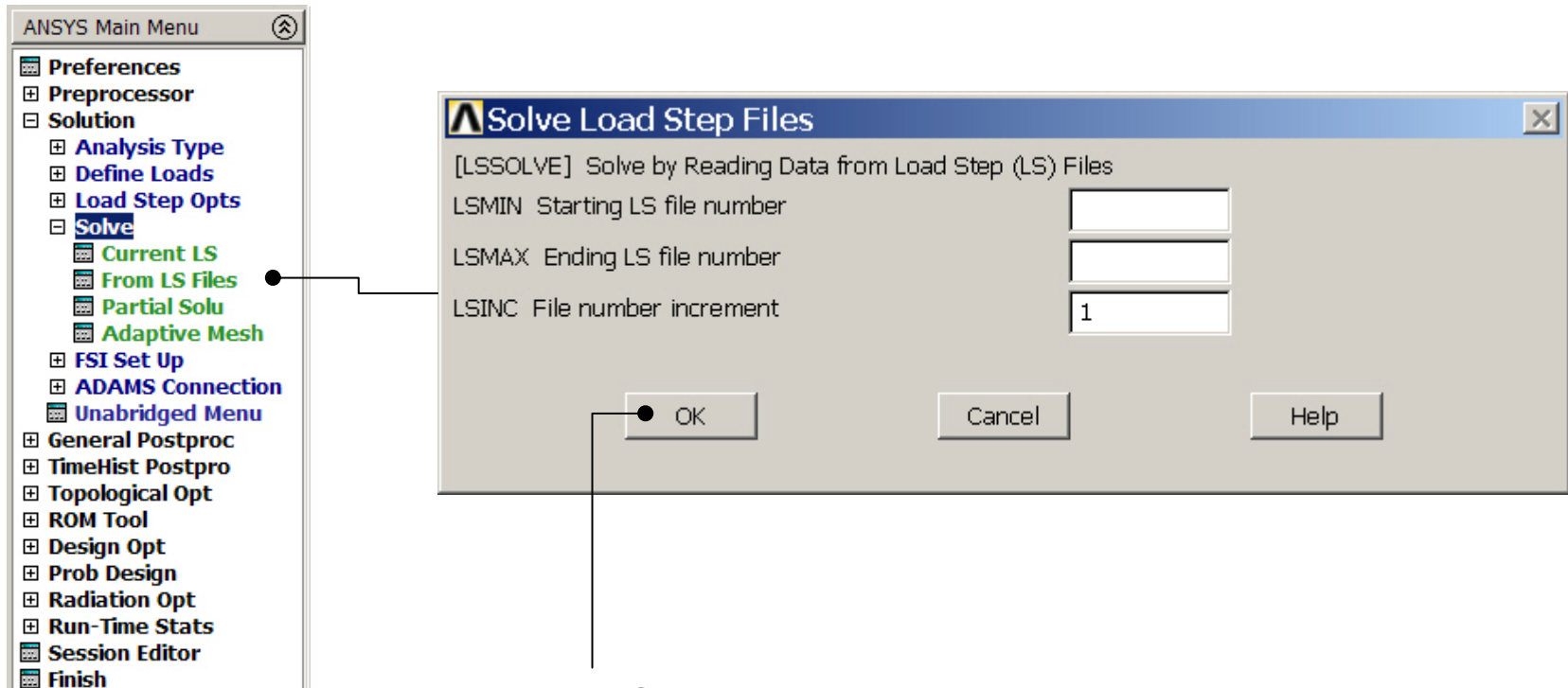
# Example – Loadstep file



Example0420

# Example - Solve

**Solution > Solve > From LS Files**

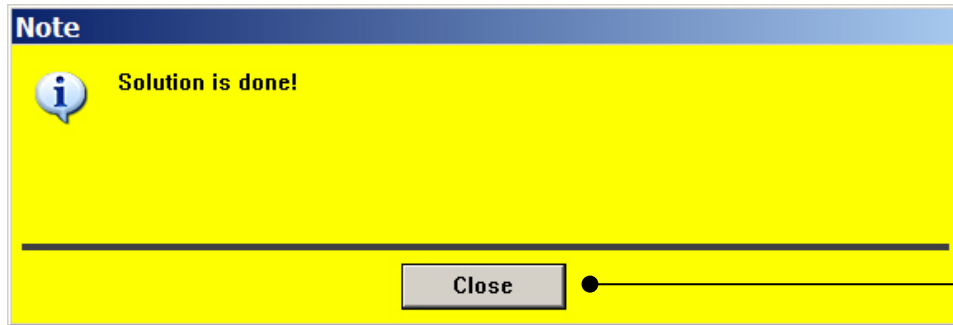


Press OK

Example0420



# Example - Solve



Press Close

# 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 is titled 'Time History Variables - file.rst' and contains a Variable List table, a Calculator, and various function buttons.

**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

MIN CONJ e^x

MAX a+ib LN 7 8 9 / CLEAR

RCL

STO LOG 4 5 6 \* ←

INS MEM SQRT

ABS ATAN x^2 1 2 3 - ENTER

INT1 IMAG

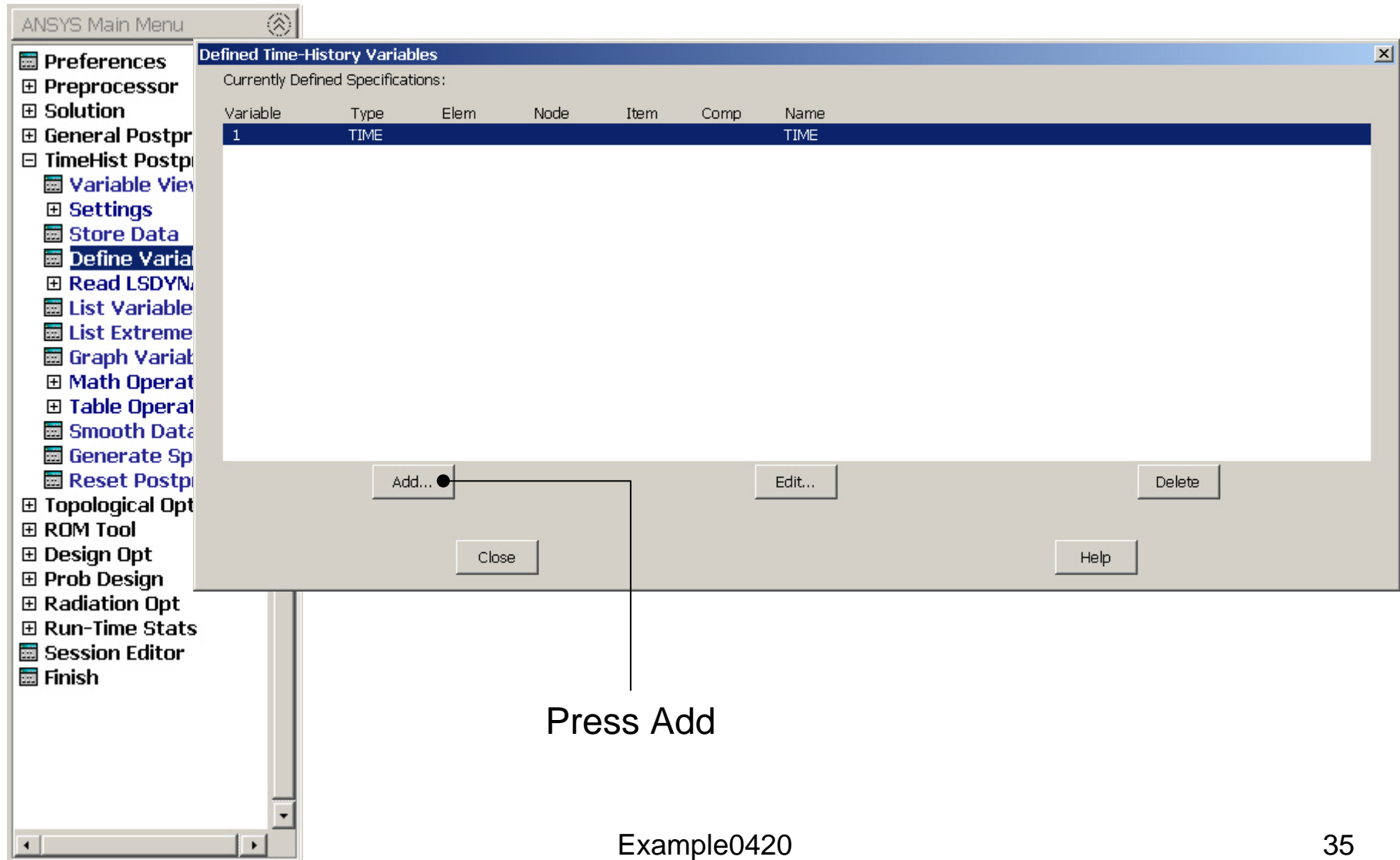
INV DERIV REAL 0 . +

Close this dialog box

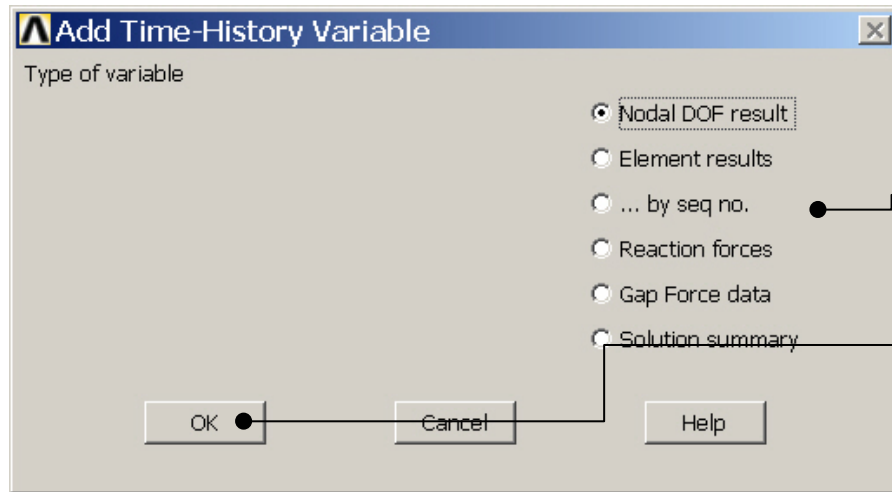
Example0420

34

# Example – Define Variables

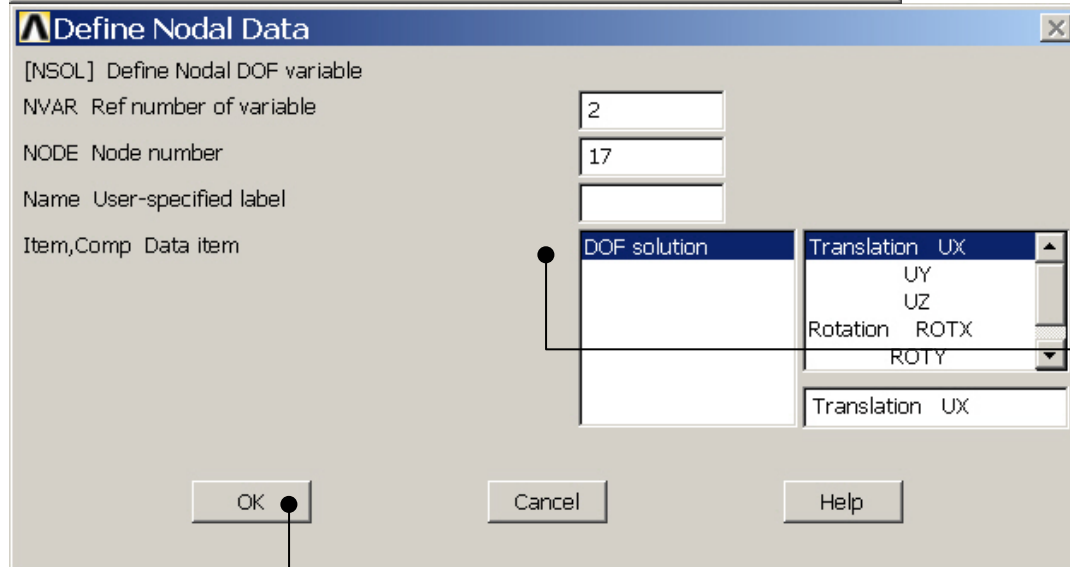


# Example – Add Time-History Var.



Select ... by  
seq no.

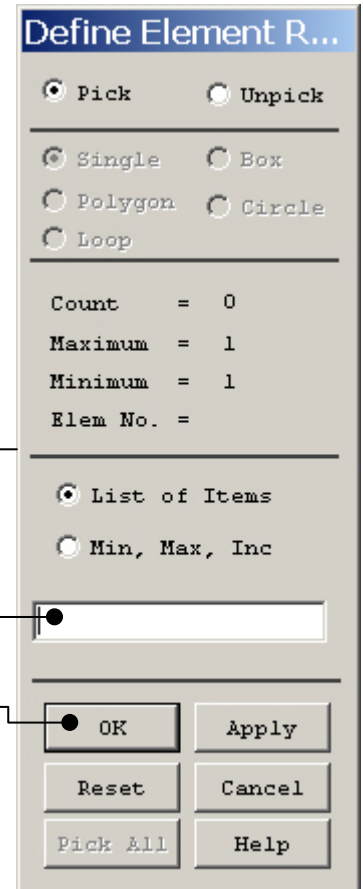
Press OK



Enter 6

Press OK

Select DOF solution  
and Translation UX



**Define Element Results by Seq No.**

[ESOL] Define Element Results Variable by Sequence No.

NVAR Ref number of variable

ELEM Element number

Name User-specified label

Item Data item

- SMISC
- NMISC
- LS
- LEPEL
- LEPTH
- LEPPL
- LEPCR
- NLIN
- LBFE

Comp Sequence number

(See Table 4.xx-3 in Elements Manual for sequence numbers.)

Optionally specify the -

[FORCE] Force results are

[SHELL] Location for shell elem

[LAYERP26] Layer results are fr

☐ Max failure crit

☒ Specified layer

Specified layer number

OK Cancel Help

**Define Element Results by Seq No.**

[ESOL] Define Element Results Variable by Sequence No.

NVAR Ref number of variable

ELEM Element number

Name User-specified label

Item Data item

- SMISC
- NMISC
- LS
- LEPEL
- LEPTH
- LEPPL
- LEPCR
- NLIN
- LBFE

Comp Sequence number

(See Table 4.xx-3 in Elements Manual for sequence numbers.)

Optionally specify the -

[FORCE] Force results are

[SHELL] Location for shell elem

[LAYERP26] Layer results are fr

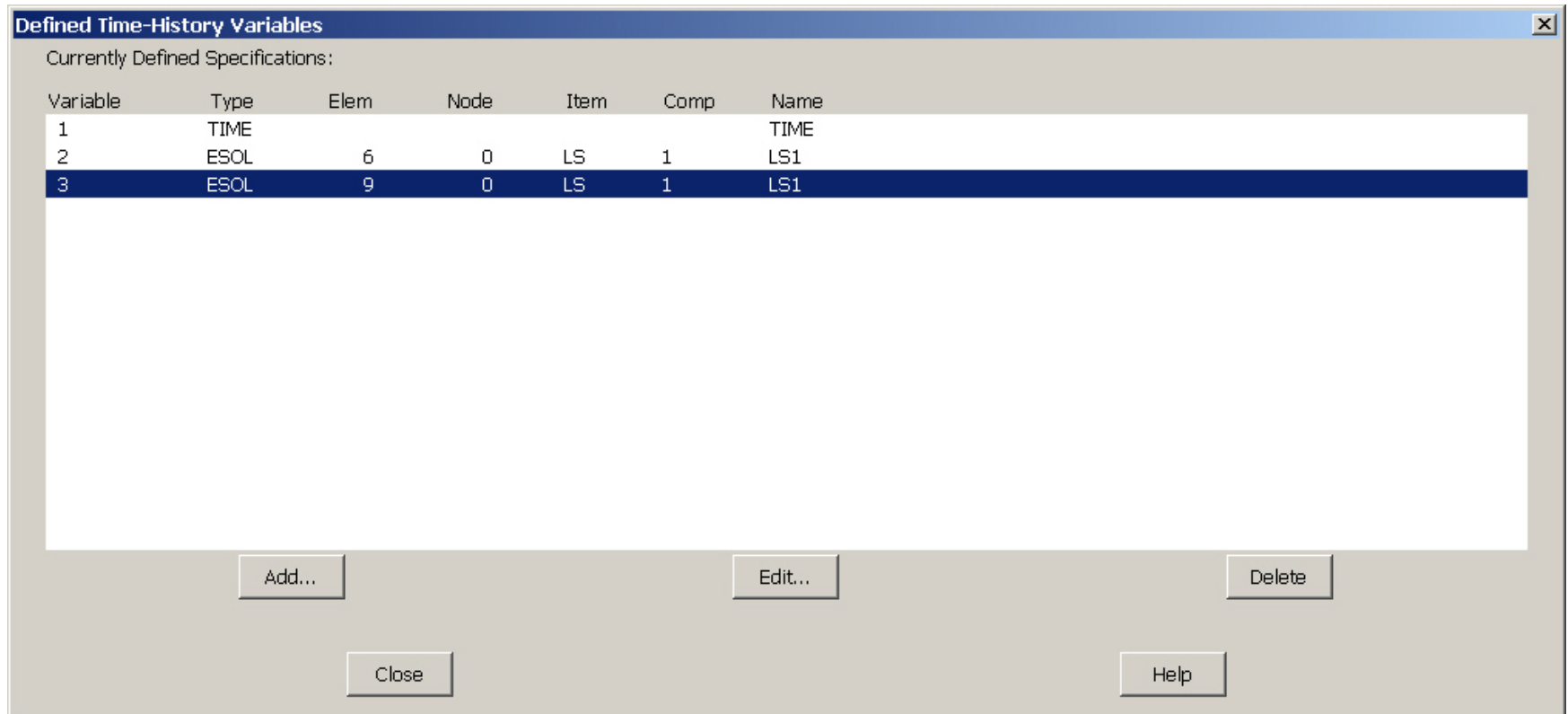
☐ Max failure crit

☒ Specified layer

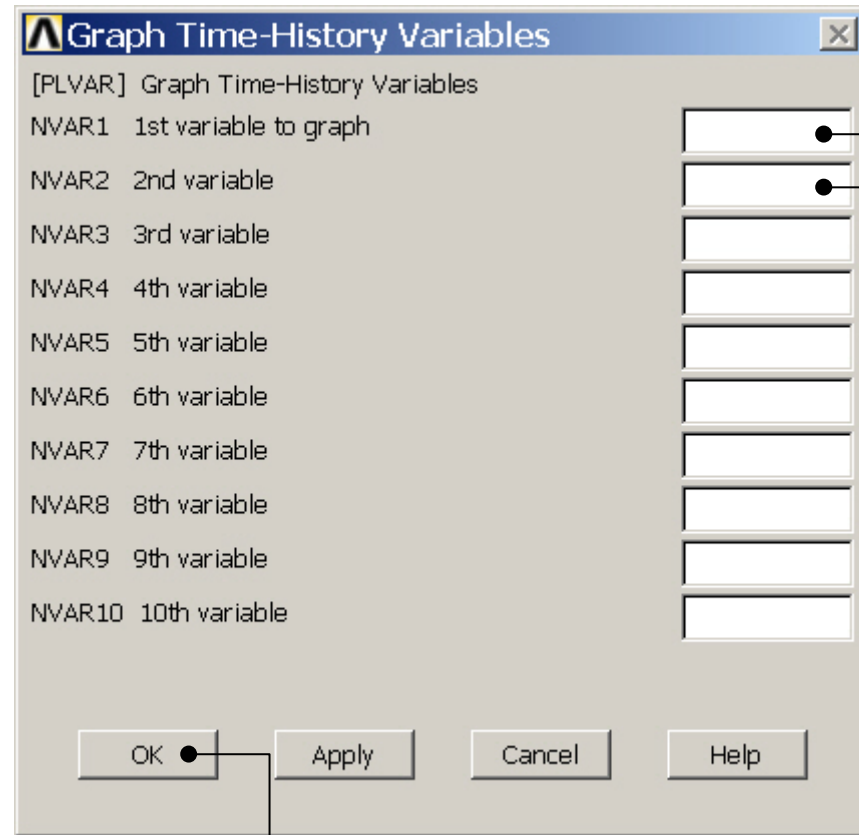
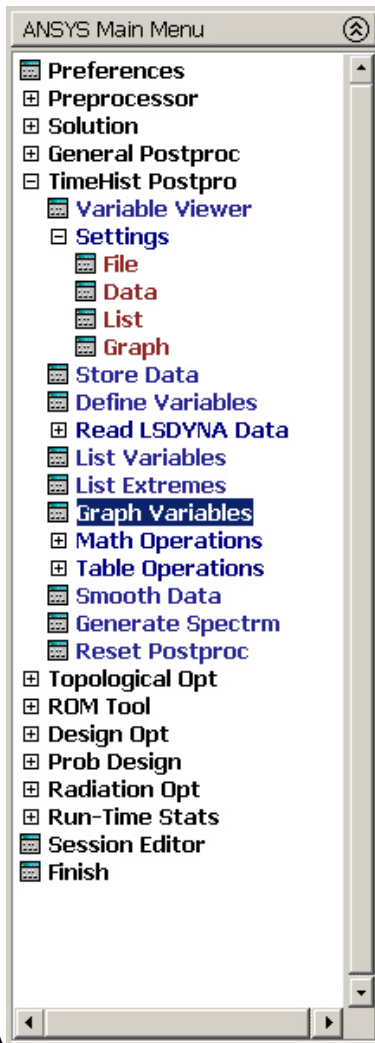
Specified layer number

OK Cancel Help

# Example – Add Time-History Var.



# Example – Graph Variables



Enter 2

Enter 3

Press OK

Example0420



# Example – Graph Variables

