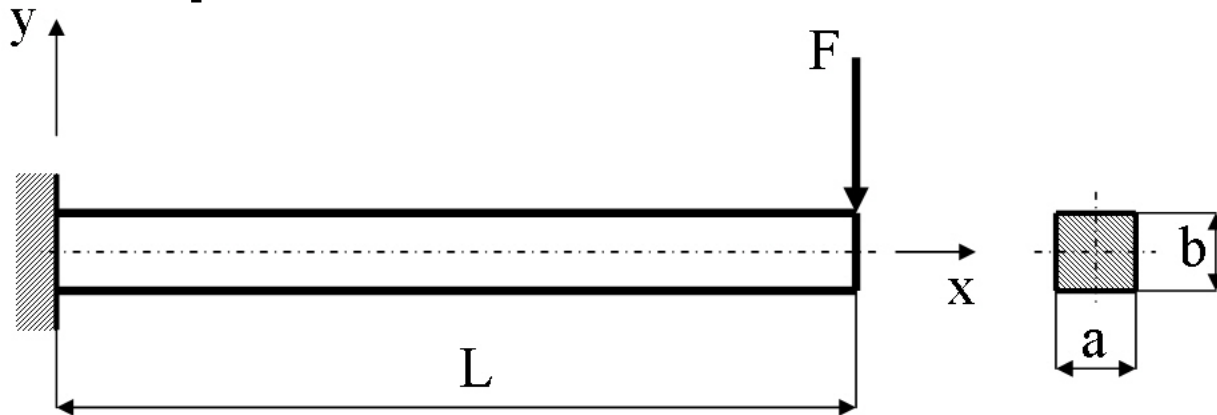


# Course in ANSYS

Example0570

# Example – Cantilever beam



## Objective:

Run the problem using different material models

## Tasks:

Run a static linear model

Run a static full nonlinear model with:

- A bilinear kinematic hardening behaviour

## Topics:

Element type, Real constants, modeling,

Plot results, output graphics, nonlinear solution control

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

$$\nu = 0.3$$

$$L = 100 \text{ mm}$$

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

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

$$\sigma_y = 355 \text{ N/mm}^2$$

$$F = 1000 \text{ N}$$

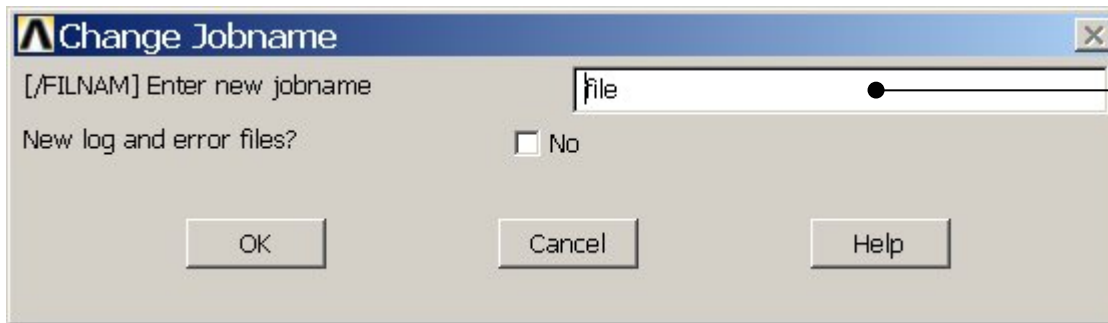
# Example - title

**Utility Menu > File > Change Jobname**

/jobname, Example0570

GUI

Command line entry

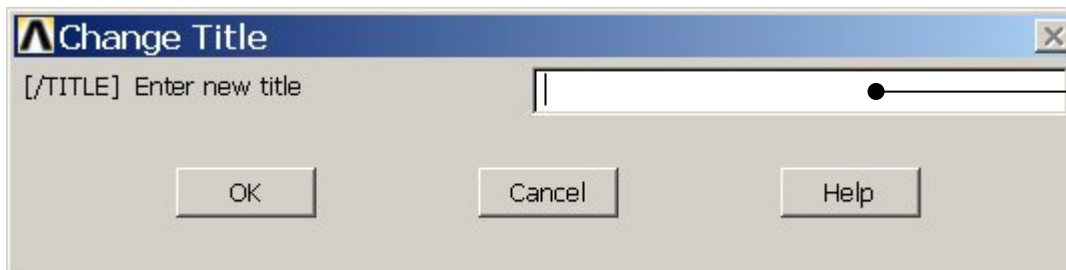


Enter: Example0570

**Utility Menu > File > Change Title**

/title, Cantilever beam

Enter: Cantilever beam



# Example - Areas

**Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions**

Create an area given by  $X=(0,100)$  and  $Y=(0,10)$

The image shows the ANSYS Main Menu on the left and the 'Create Rectangle by Dimensions' dialog box in the center. The dialog box has a title bar with the ANSYS logo and the text 'Create Rectangle by Dimensions'. Below the title bar, it says '[RECTNG] Create Rectangle by Dimensions'. There are two rows of input fields: 'X1,X2 X-coordinates' and 'Y1,Y2 Y-coordinates'. Each row has two empty text boxes. Below the input fields are four buttons: 'OK', 'Apply', 'Cancel', and 'Help'. There are four callout lines with dots pointing to the input fields and the 'OK' button. The callouts are: 'Enter 0 or leave empty' (pointing to the first X input), 'Enter 100' (pointing to the second X input), 'Enter 0 or leave empty' (pointing to the first Y input), and 'Enter 10' (pointing to the second Y input). A fifth callout line points to the 'OK' button with the text 'Press OK'.

Enter 0 or leave empty

Enter 100

Enter 0 or leave empty

Enter 10

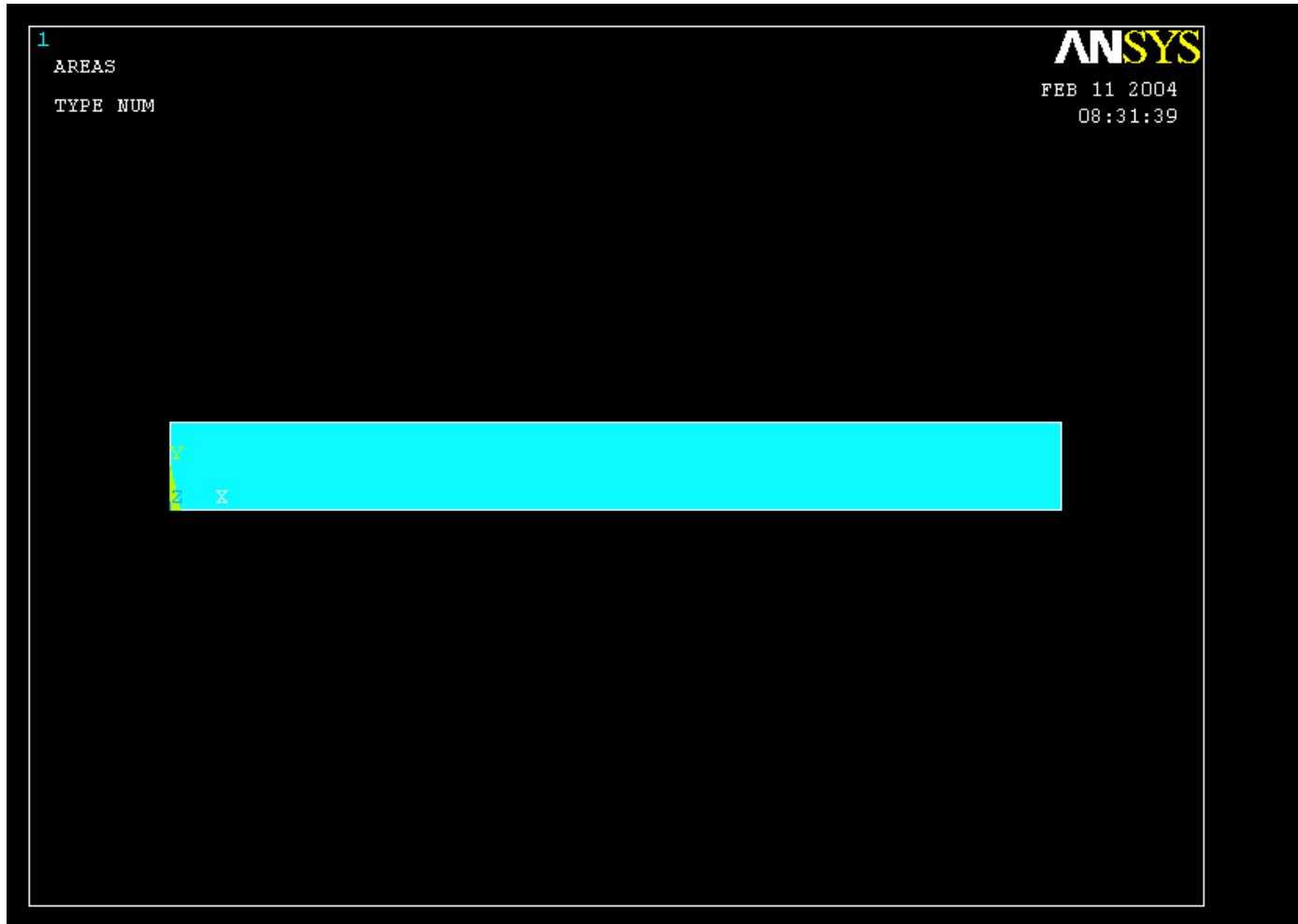
Press OK

Note: Keypoints (4 kp's) and lines (4 lines) are automatically generated (also numbered automatically)

Example0570

Computational mechanics, AAU, Esbjerg

# Example - Area



# Example - Operate

**Preprocessor > Modeling > Operate > Extrude > Areas > Along Normal**

Create a volume by extruding the area 10 along its surface normal vector

ANSYS Main Menu

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
  - Sections
  - Modeling
    - Create
    - Operate
      - Extrude
        - Elem Ext Opts
        - Areas
          - Along Normal
          - By XYZ Offset
          - About Axis
          - Along Lines
        - Lines
        - Keypoints
        - Extend Line
        - Booleans
        - Scale
        - Calc Geom Items
      - Move / Modify
      - Copy
      - Reflect
      - Check Geom
      - Delete
      - Cyclic Sector
      - Genl plane strn
      - Update Geom
    - Meshing
    - Checking Ctrl
    - Numbering Ctrl
    - Archive Model

Extrude Area by N...

☒ Pick ☐ Unpick

☒ Single ☐ Box

☐ Polygon ☐ Circle

☐ Loop

Count = 0

Maximum = 1

Minimum = 1

Area No. =

☒ List of Items

☐ Min, Max, Inc

OK Apply Cancel Help

Extrude Area along Normal

[VOFFST] Extrude Area along Normal

NAREA Area to be extruded 1

DIST Length of extrusion 10

KINC Keypoint increment

OK Apply Cancel Help

Press OK

Enter 10

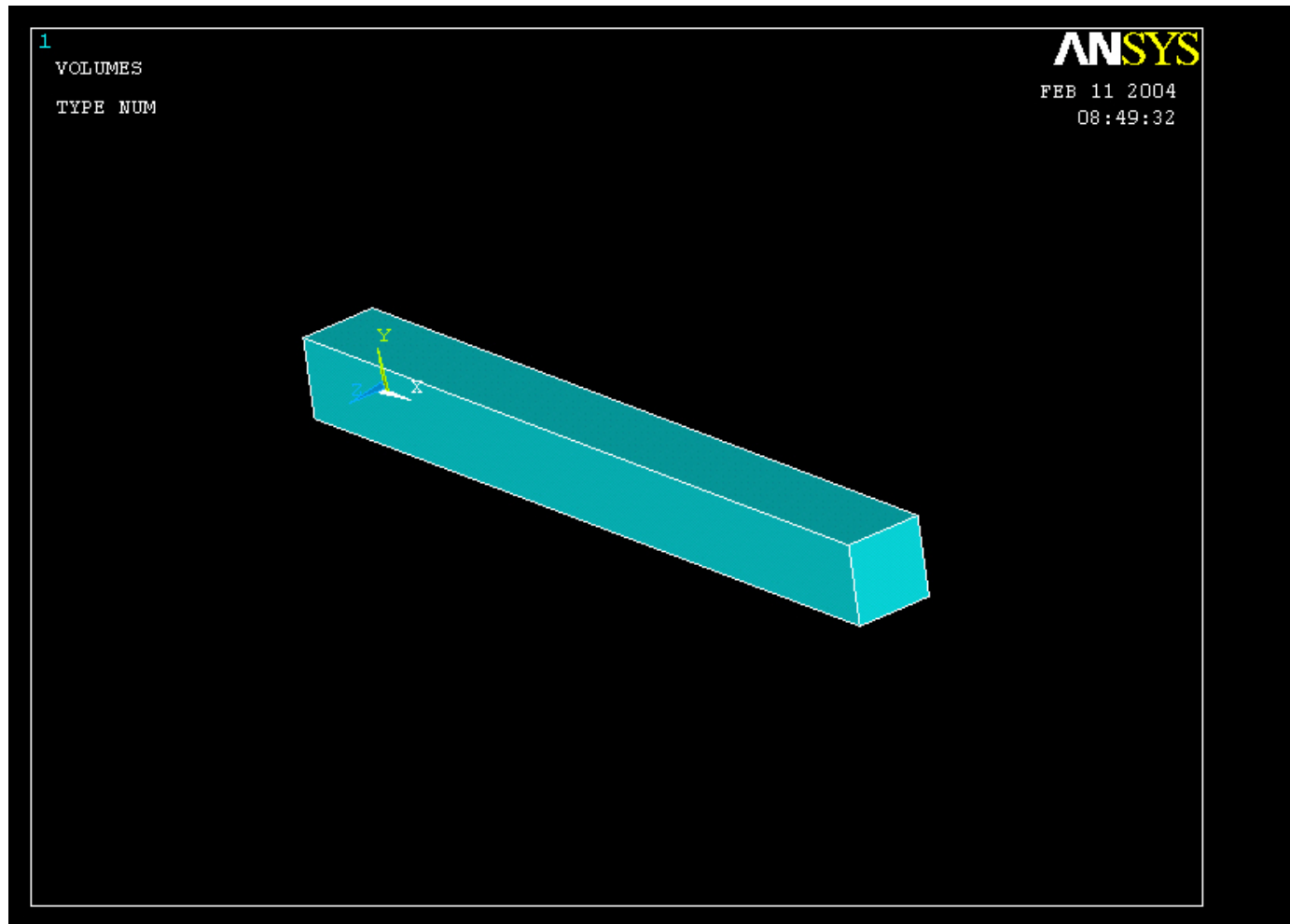
Note: Bottom left corner of ANSYS GUI

[VOFFST] Pick or enter the area to be extruded

Example0570

ics, AAU, Esbjerg

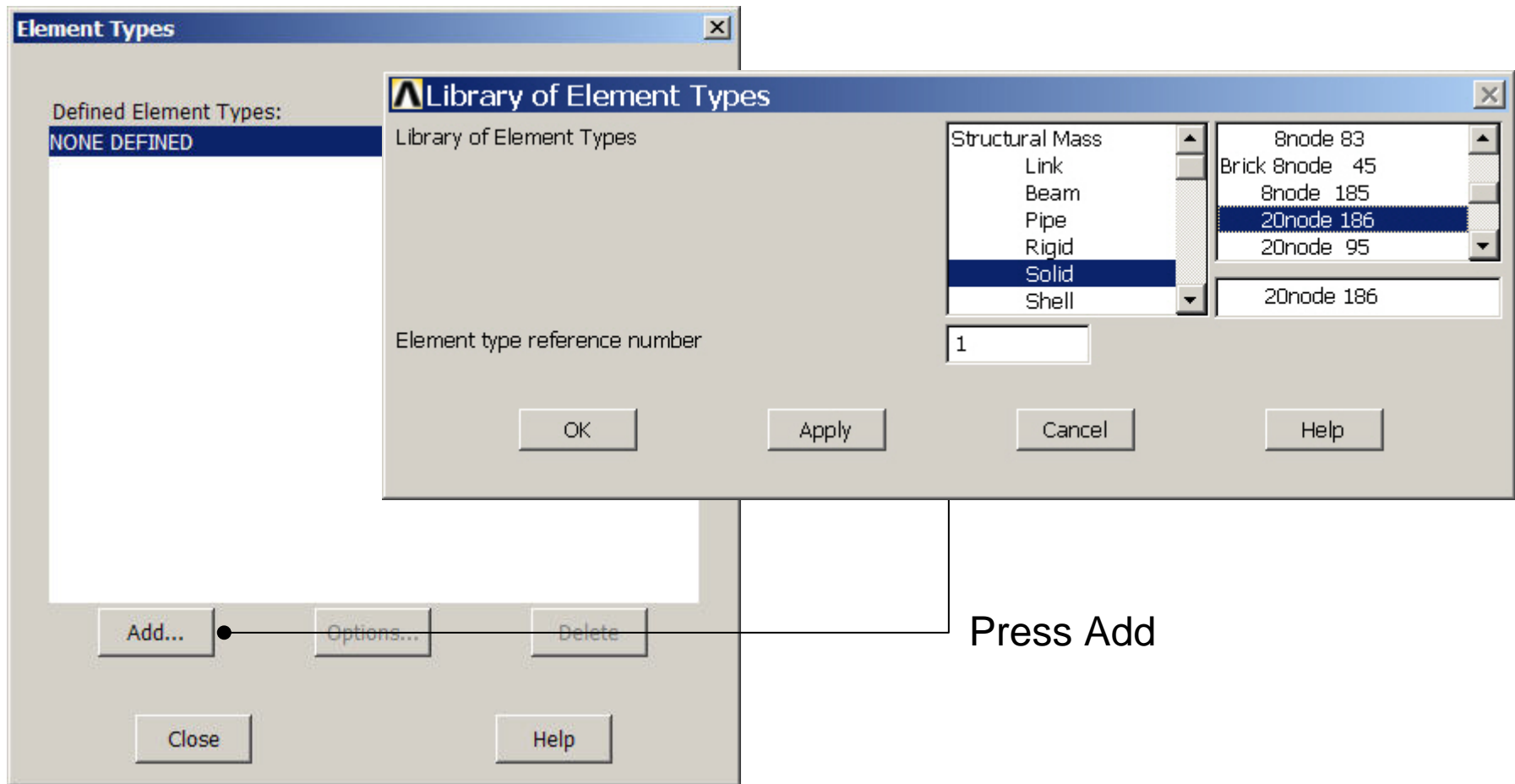
# Example – Mouse rotate



Rotate by holding the Ctrl key down while using the right hand mouse button

# Example – Element Type

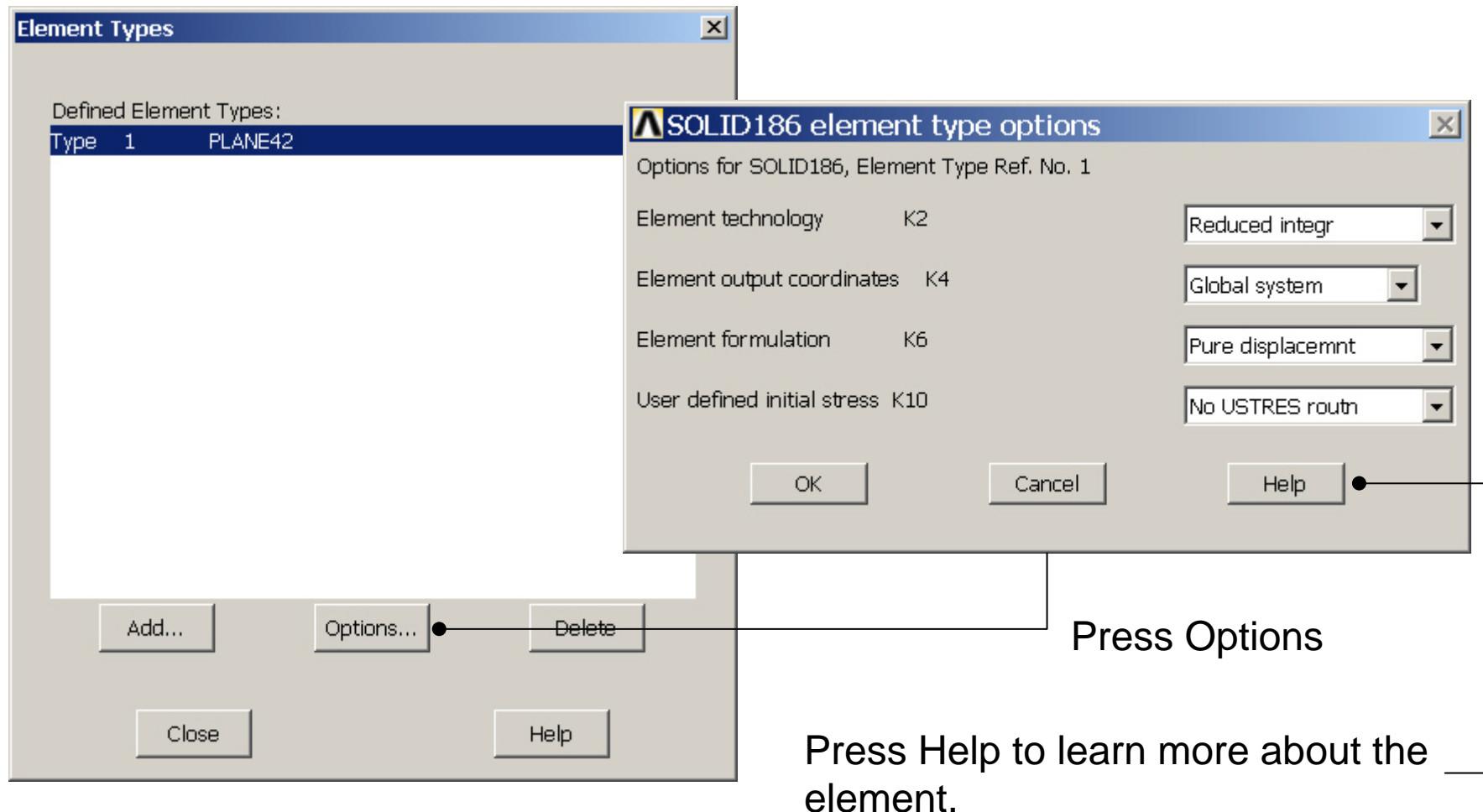
Preprocessor > Element Type > Add/Edit/Delete





# Example - Element Type

Preprocessor > Element Type > Add/Edit/Delete

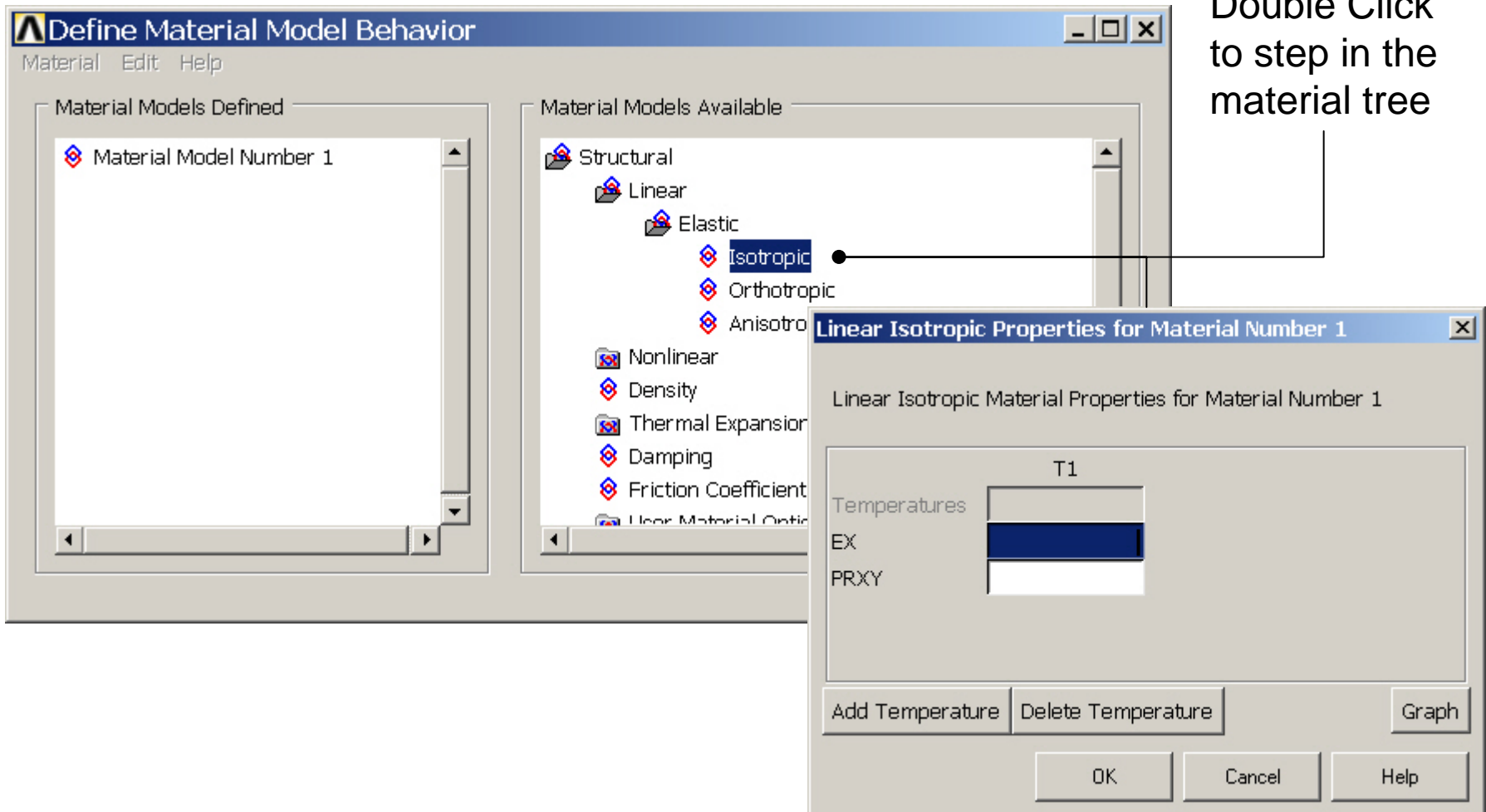


# Example – Real Constants

No Real Constants are necessary for pure 3D solid models!

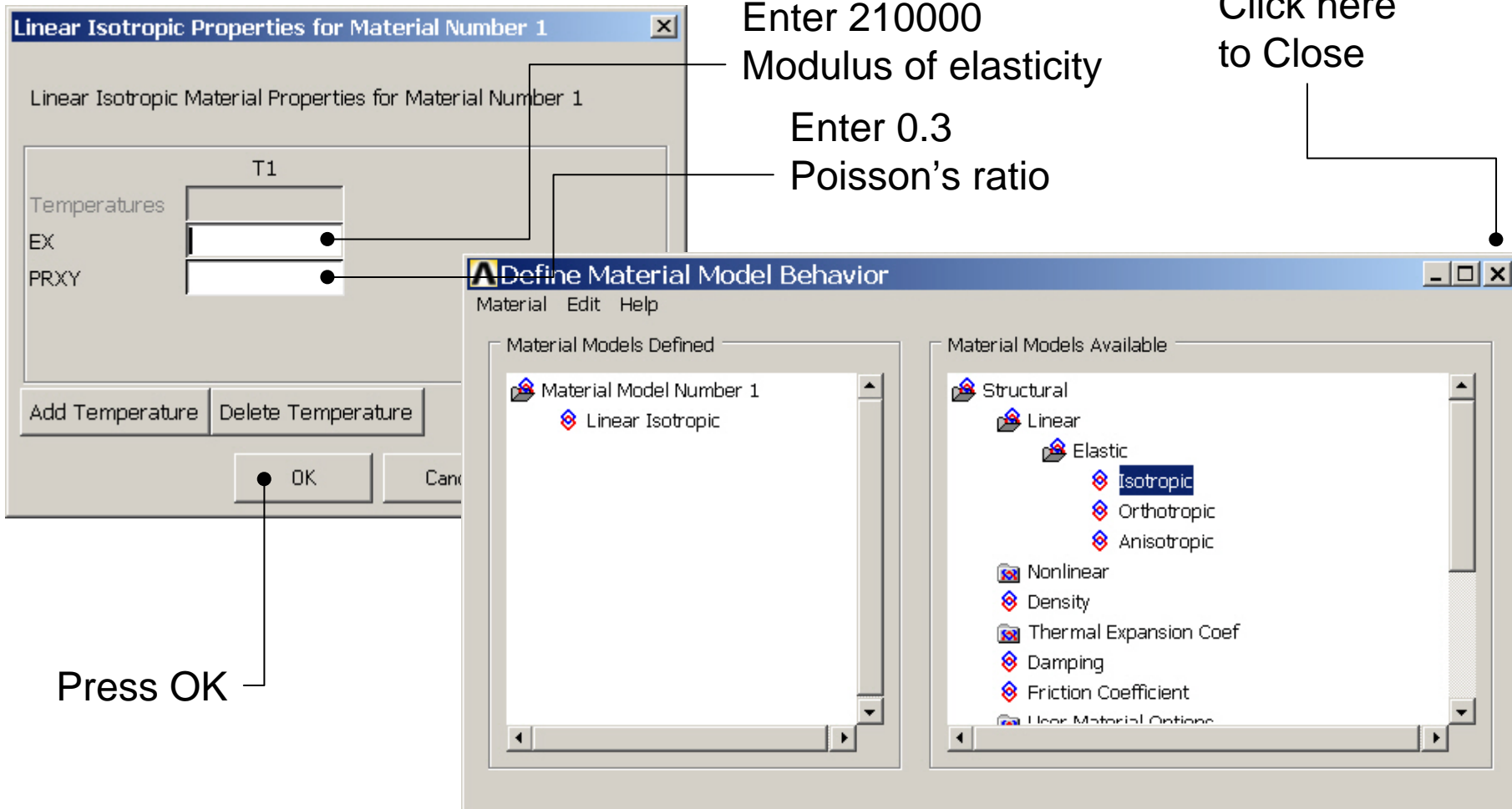
# Example - Material Properties

Preprocessor > Material Props > Material Models



# Example - Material Properties

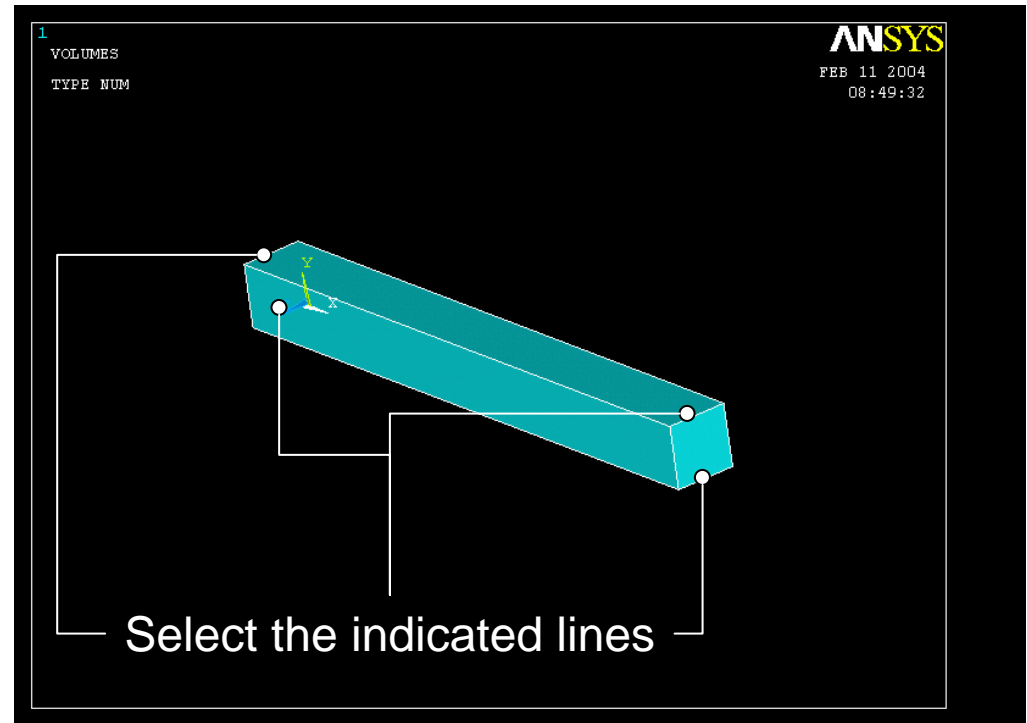
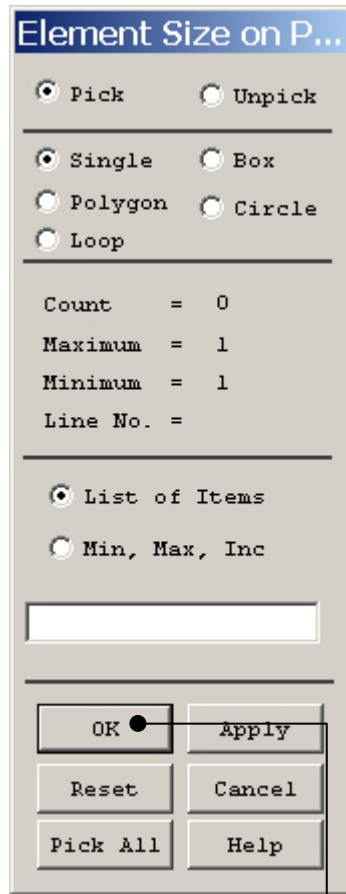
Preprocessor > Material Props > Material Models



# Example - Meshing

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

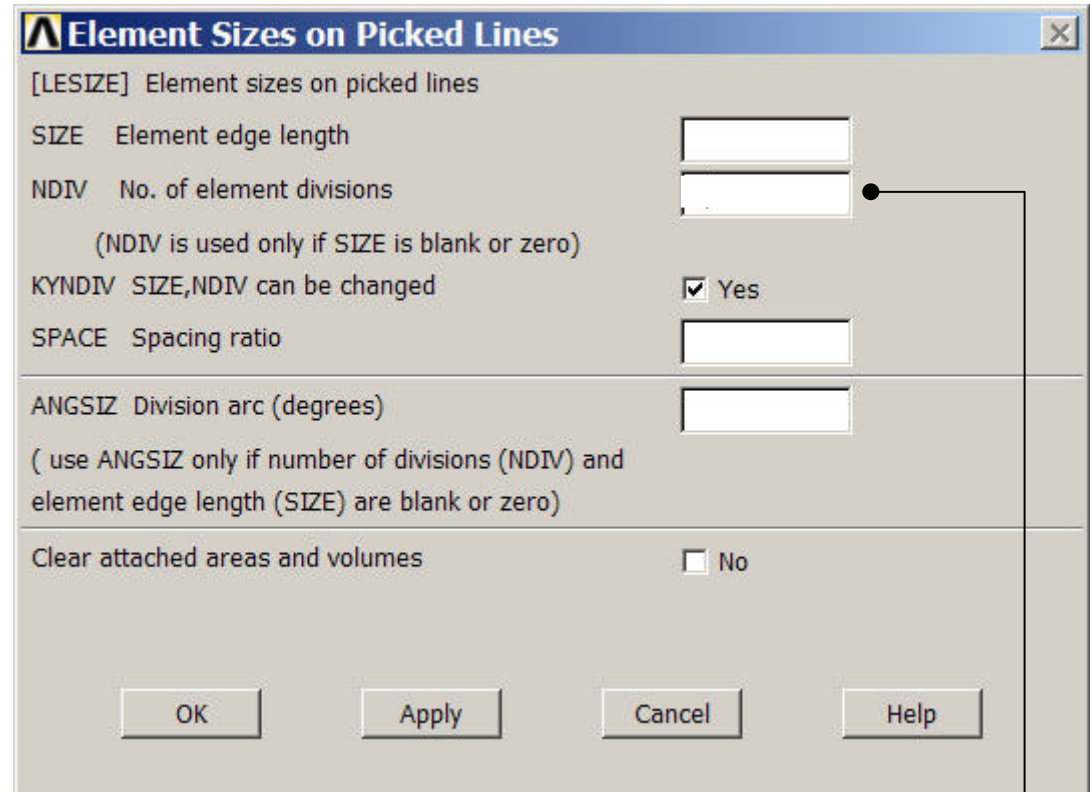
Select/Pick  
Lines to  
specify  
mesh size  
for



Press OK when finish with selection

# Example - Meshing

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

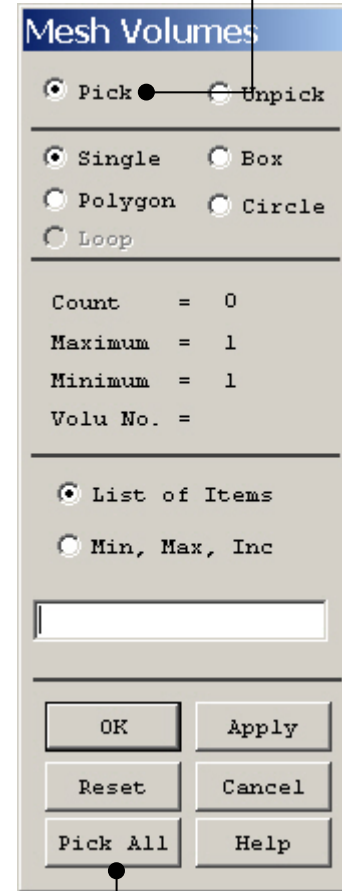


Enter 2

Press OK when finish with selection

# Example - Meshing

Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 or 6 sided

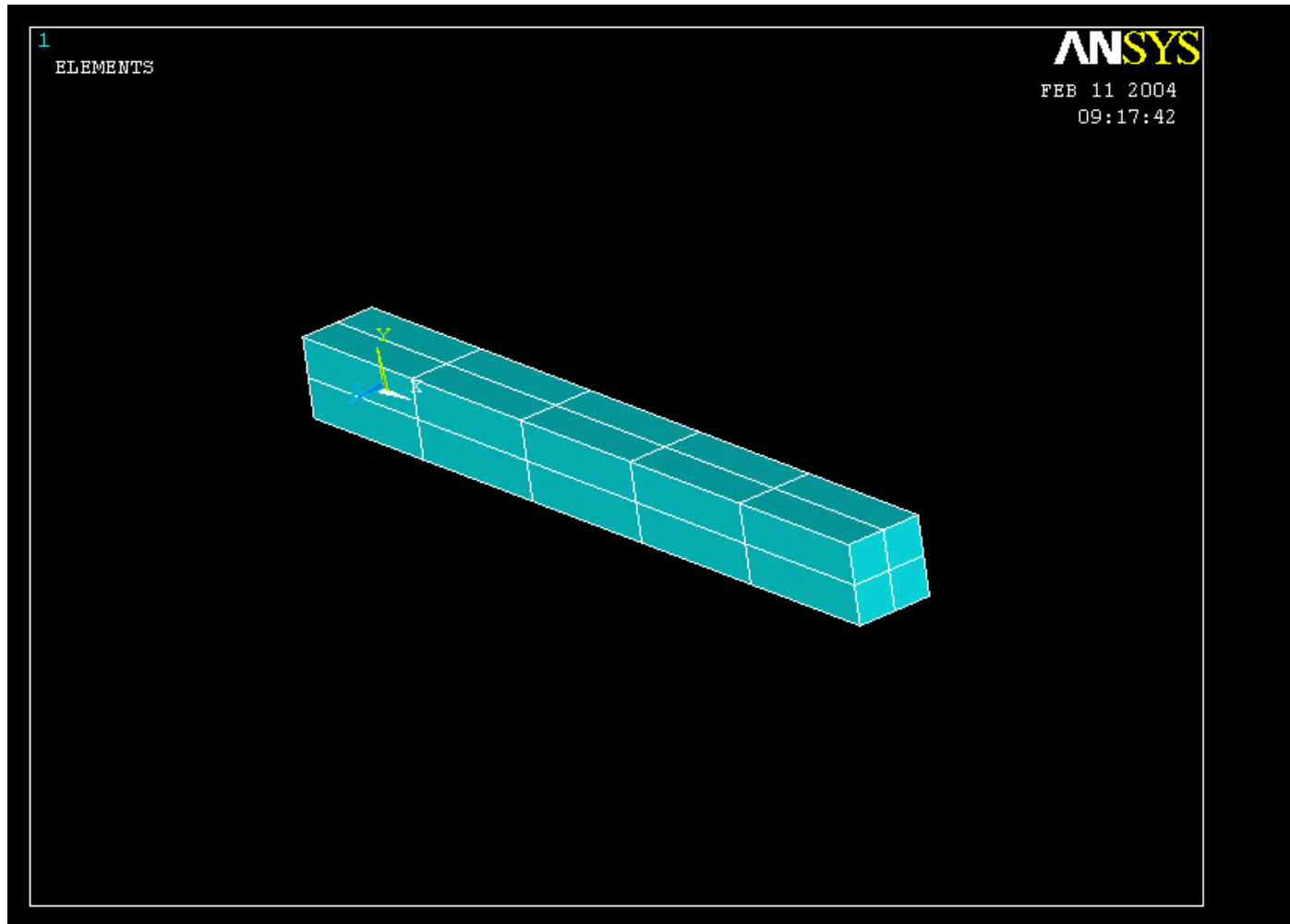


Select individual volumes to be meshed

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

Select all volumes defined to be meshed

# Example – 3D Mesh

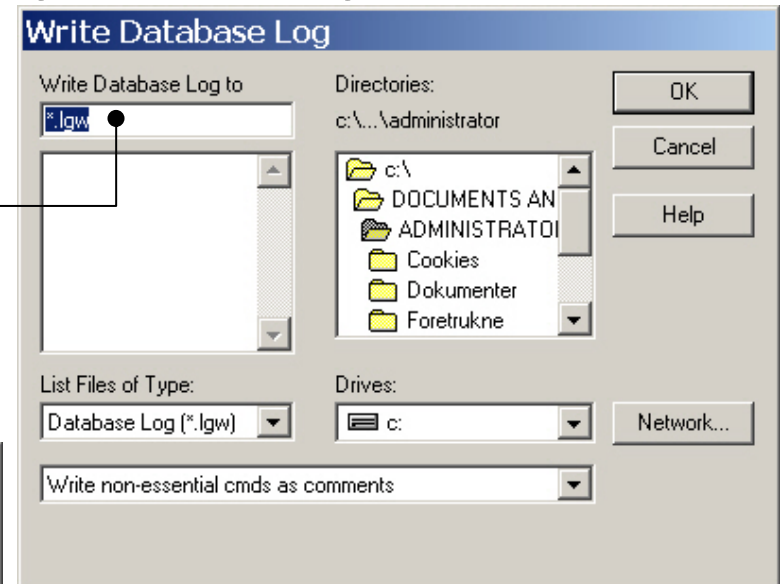




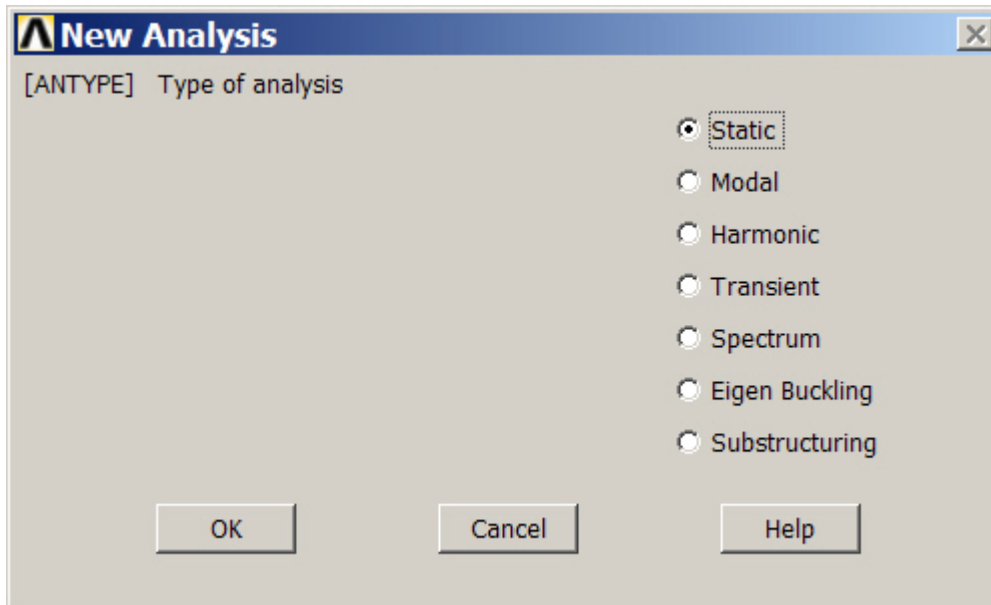
# Example – Analysis Type

**File > Write DB log file**

Enter “example0570.lgw”

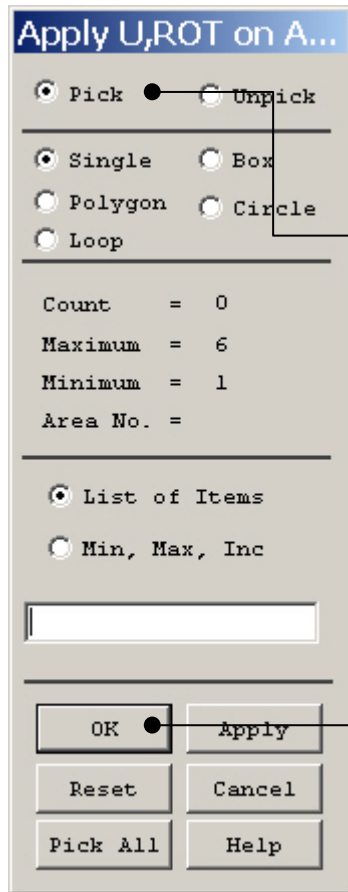


**Solution > Analysis Type > New Analysis**



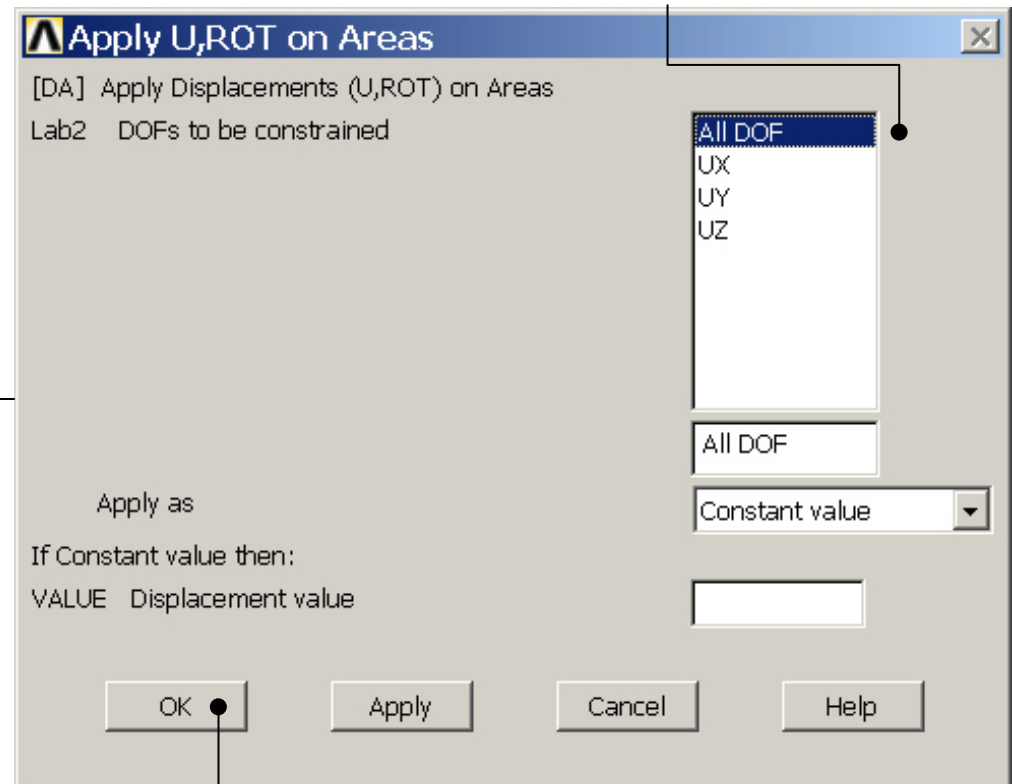
# Example – Define Loads

**Solution > Define Loads > Apply > Structural > Displacement > On Areas**



Select Area  
A6 or the left  
end area

Select All DOF to fix/clamp the beam

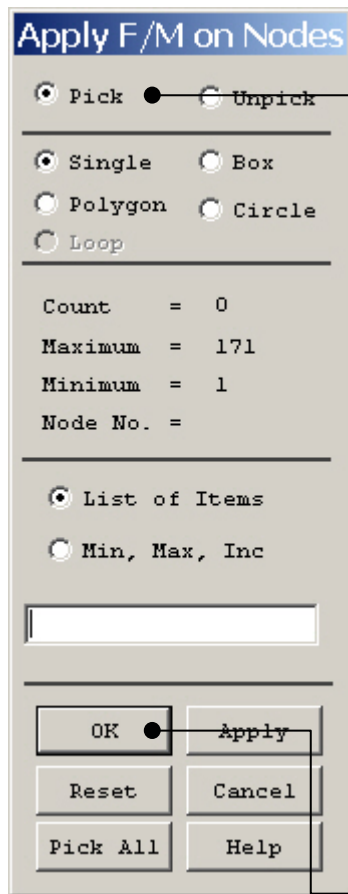


Press OK

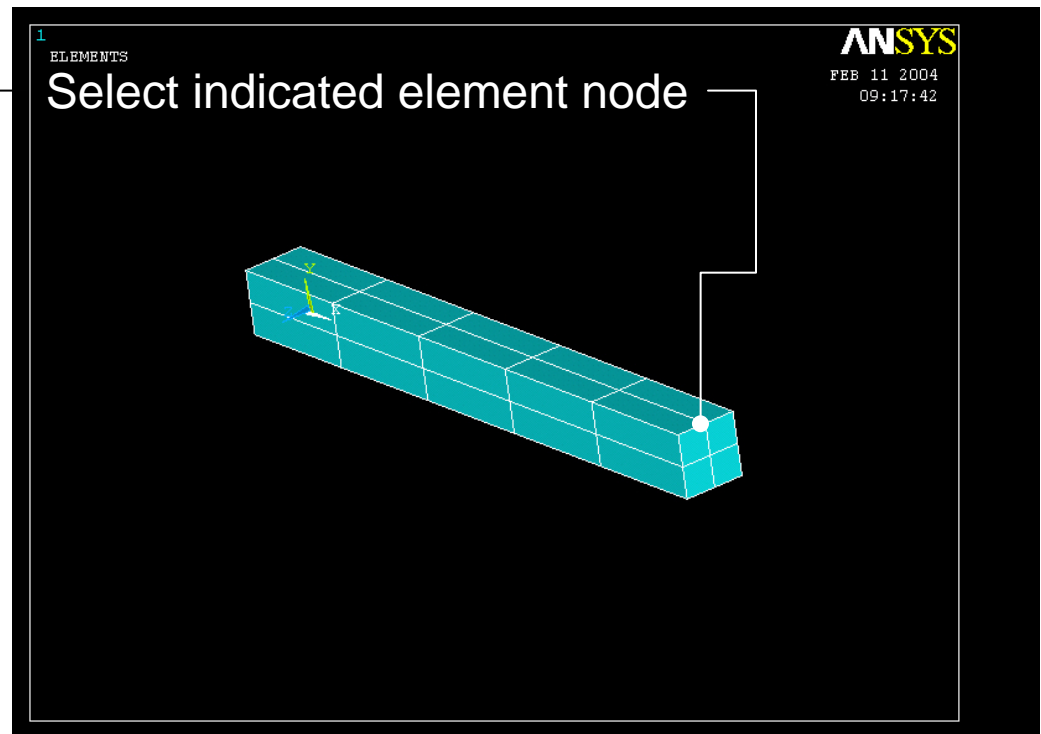
# Example – Define Loads

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

Note: If the model is remeshed all loads will be deleted with the element nodes

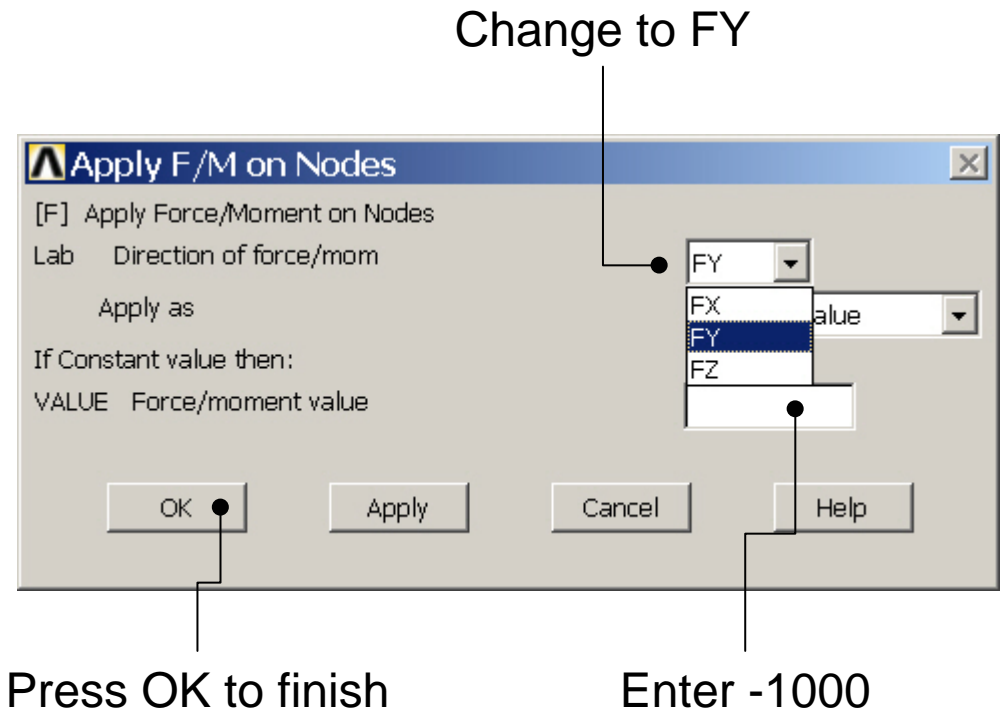


Press OK

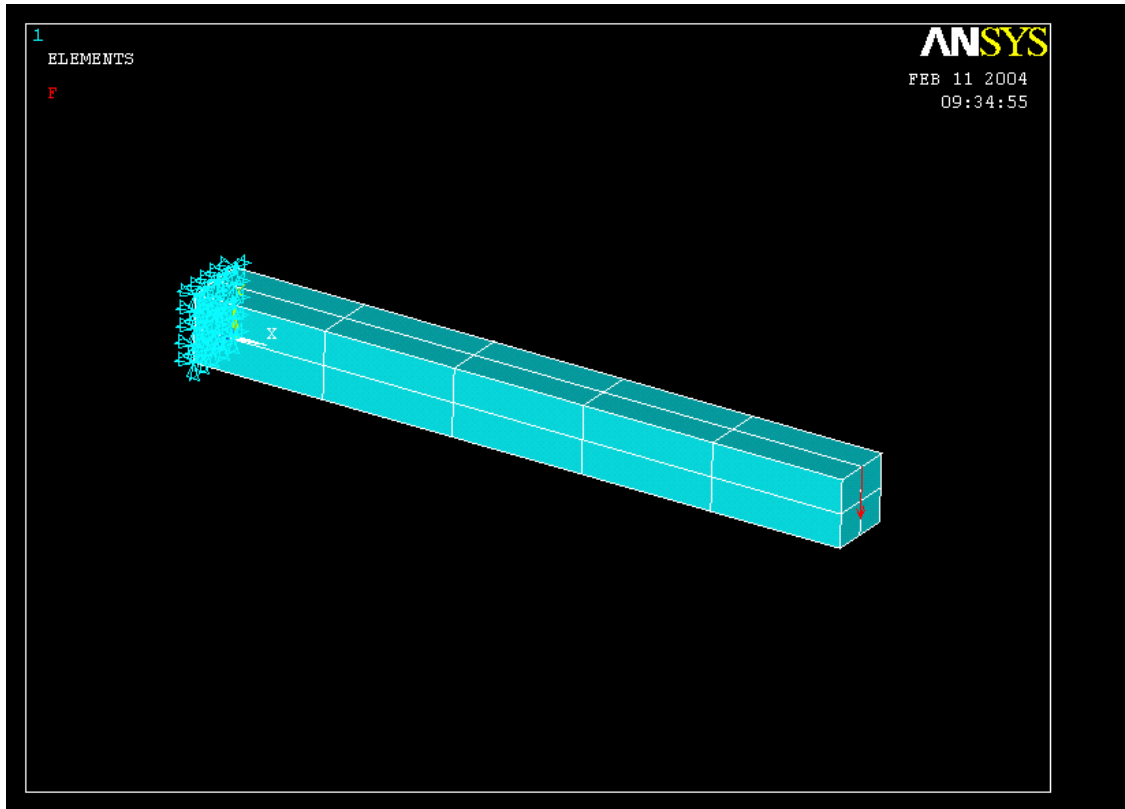


# Example – Define Loads

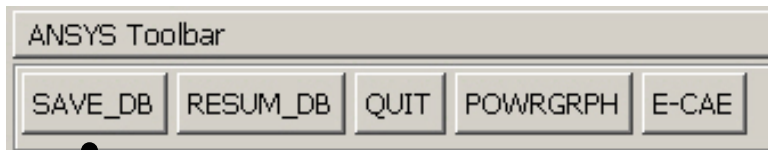
**Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes**



# Example - Save



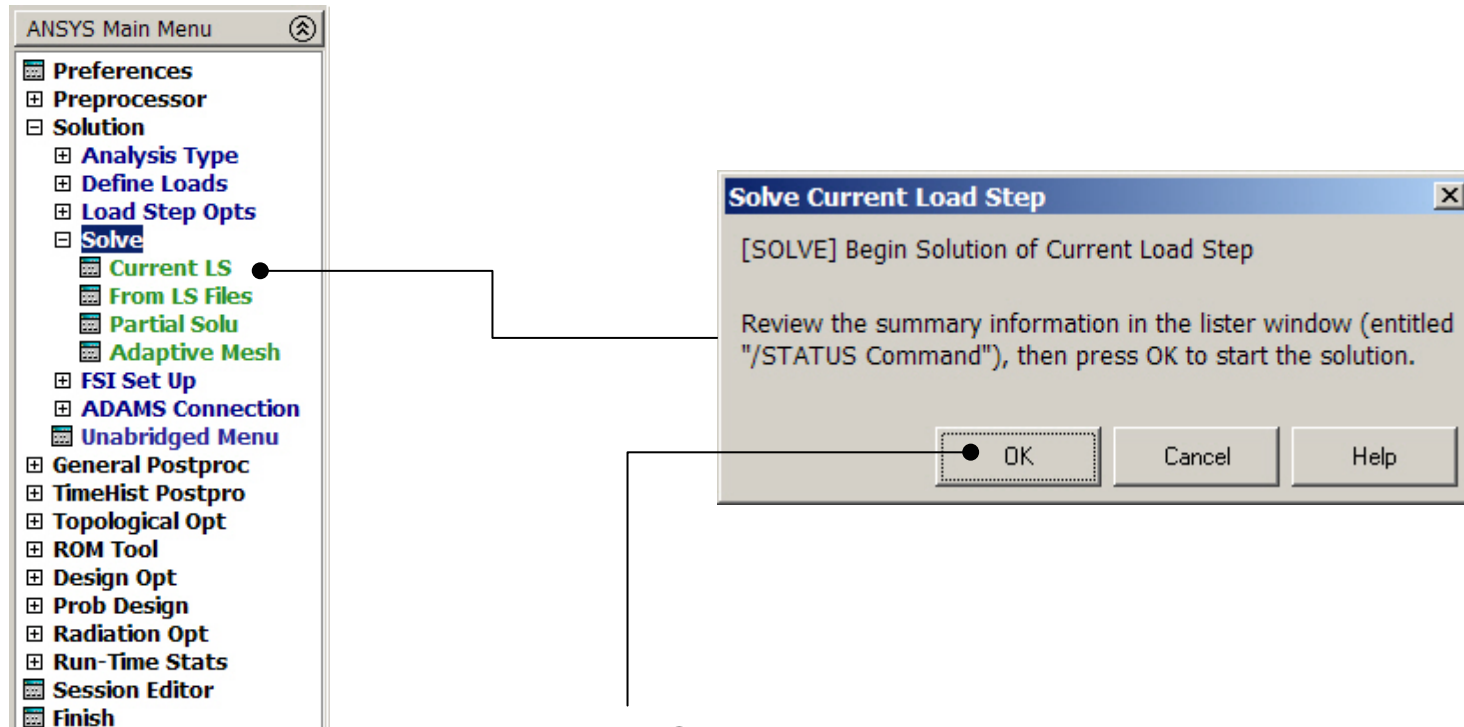
Display of Analysis model



Save the model

# Example - Solve

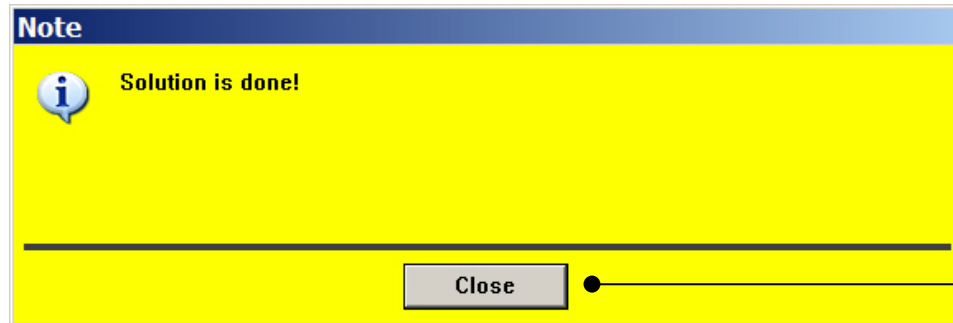
**Solution > Solve > Current LS**



Press OK

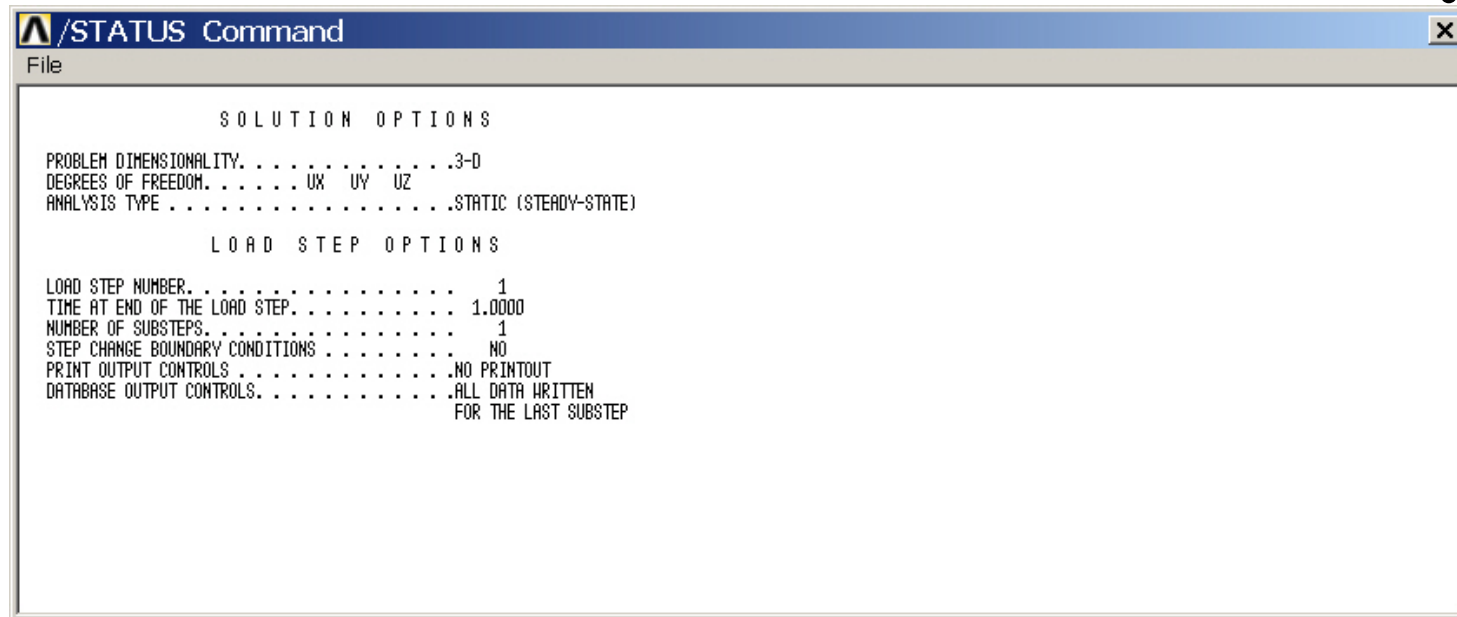
Example0570

# Example - Solve



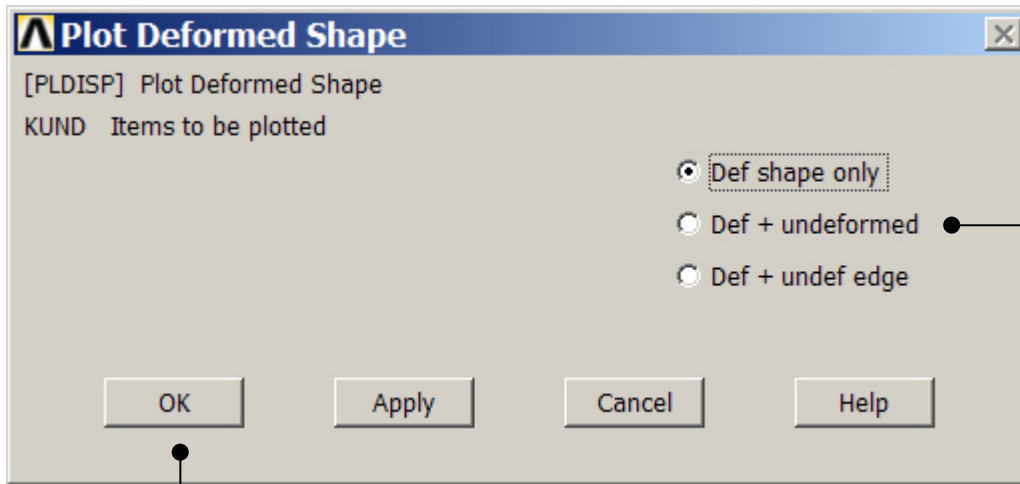
Press Close

Press here  
to Close



# Example - PostProcessing

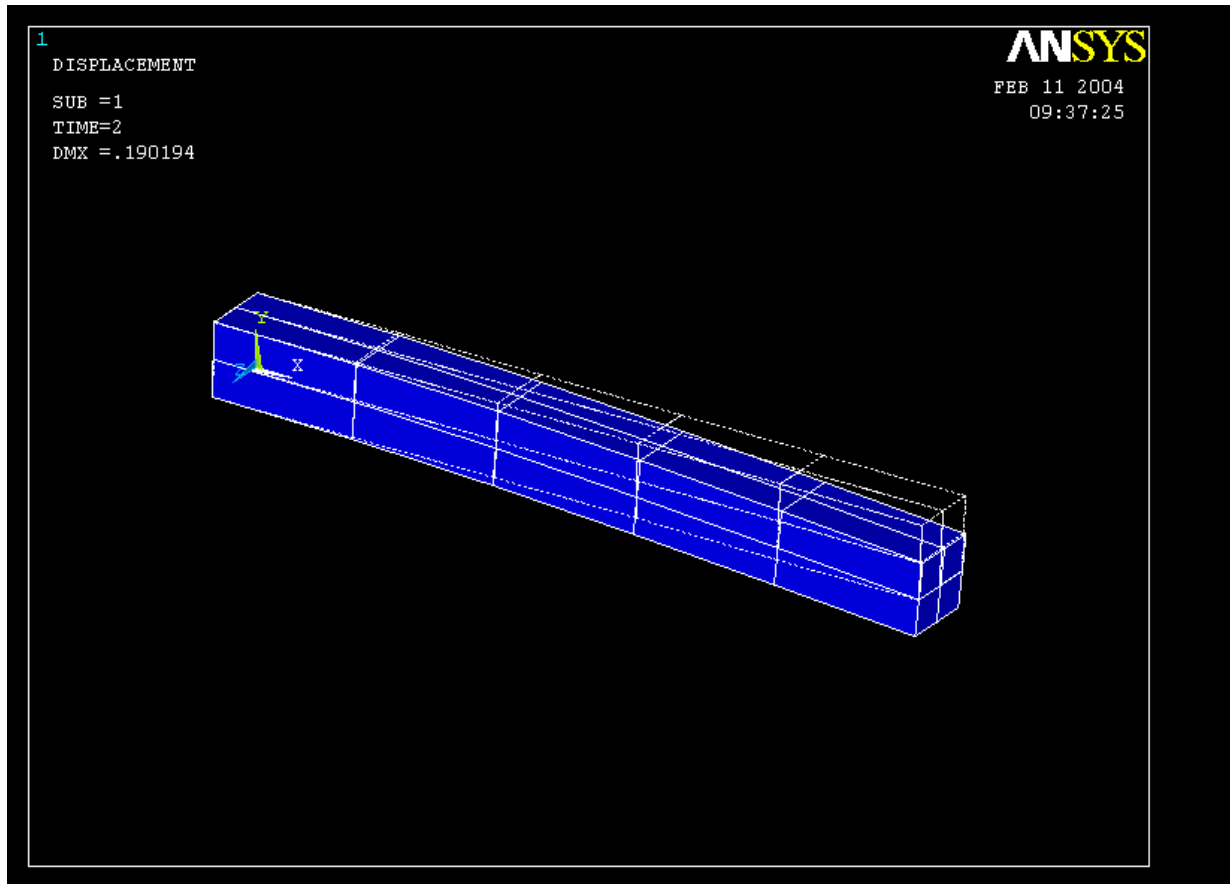
General Postproc > Plot Results > Deformed Shape



Select "Def+undeformed"  
and Press OK

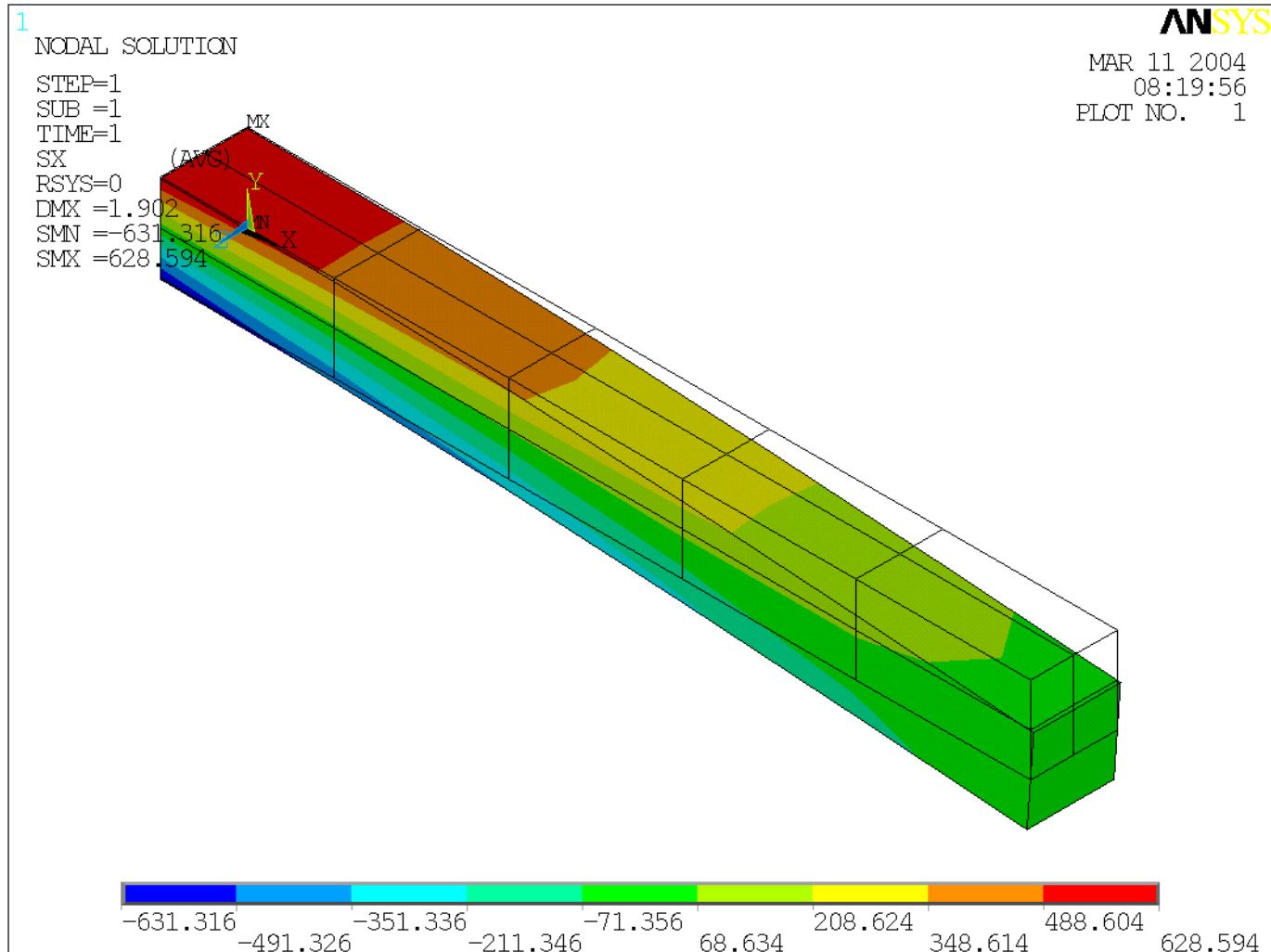


# Example - PostProcessing

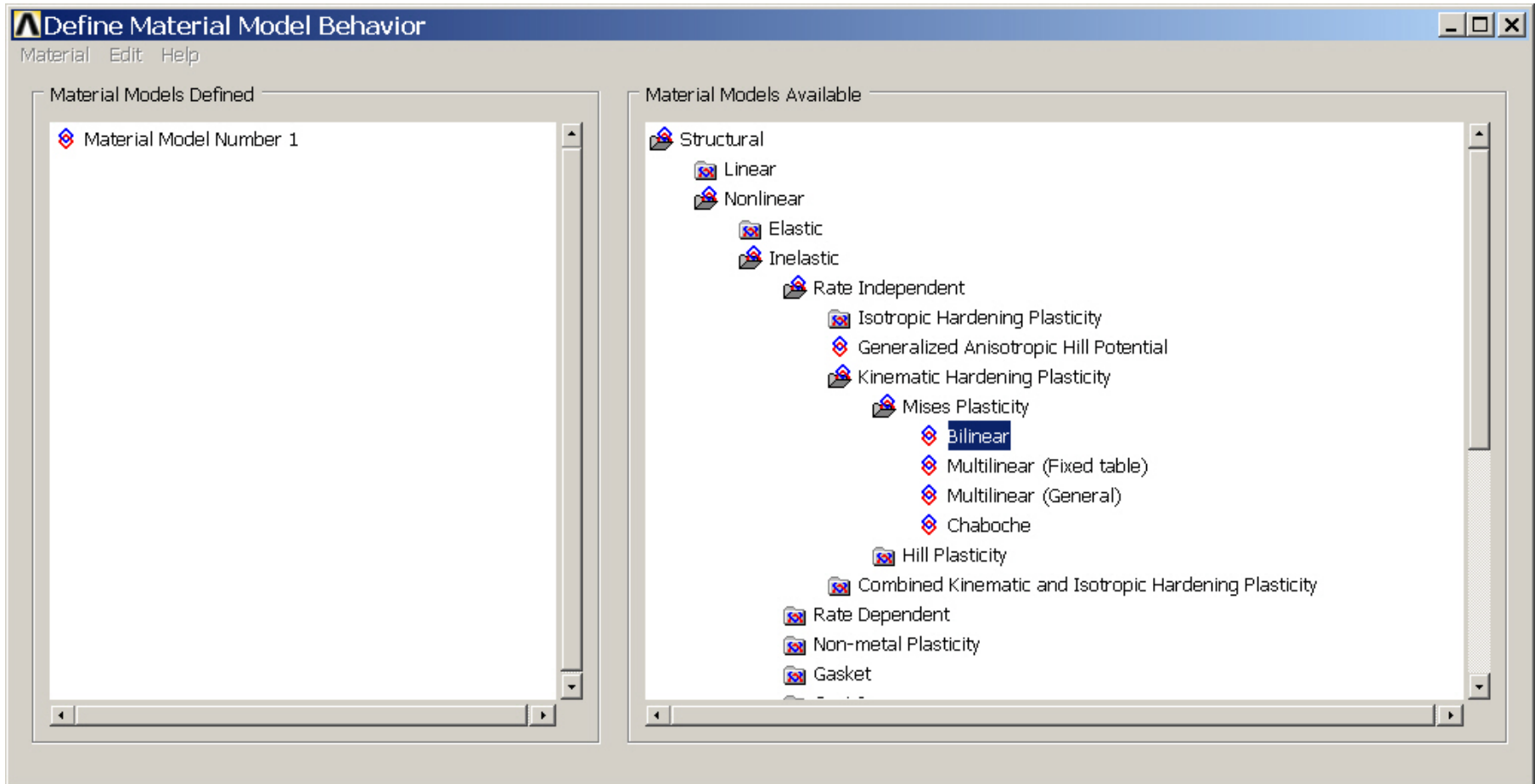


Read Maximum displacement: DMX

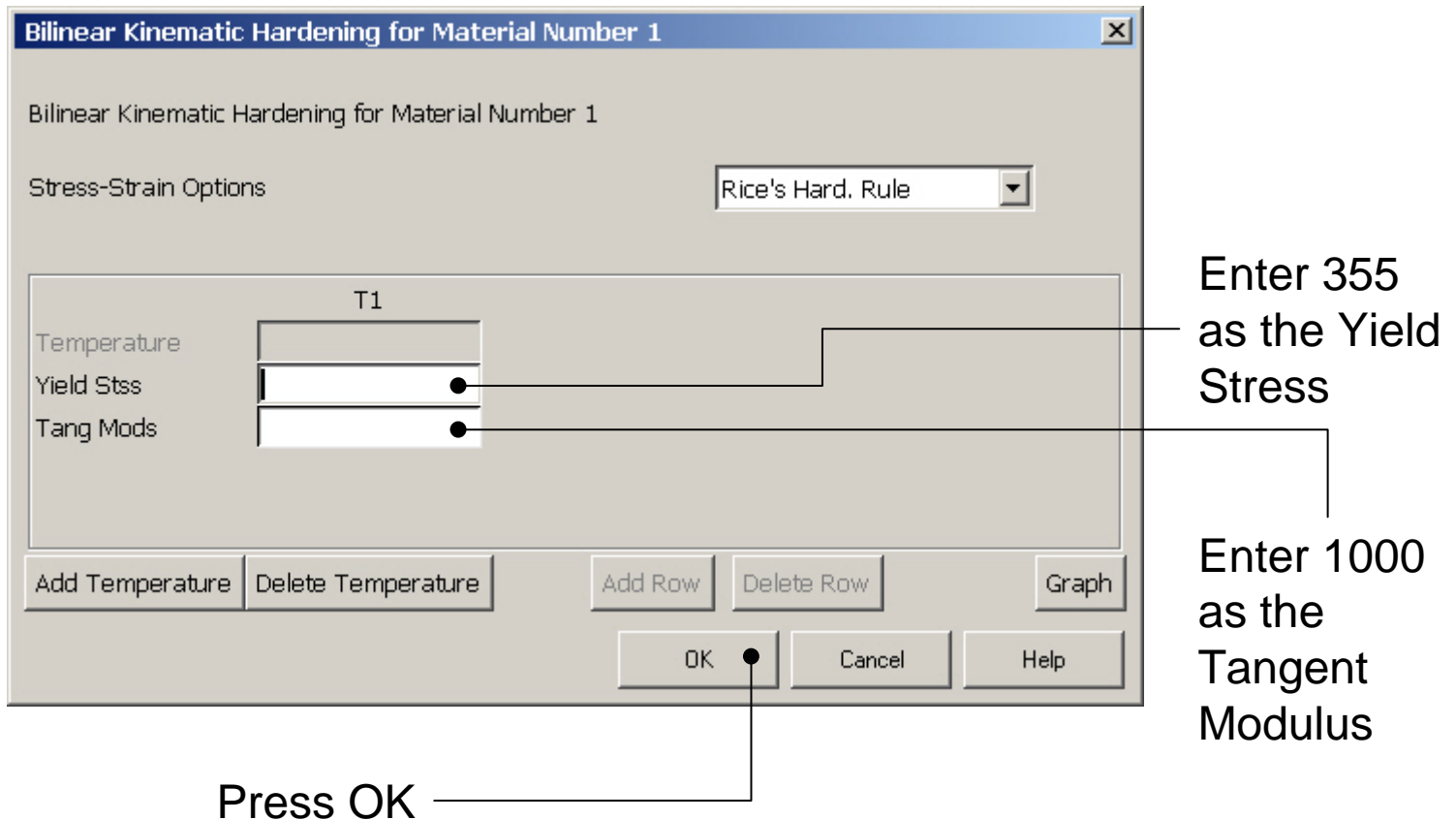
# Example – Linear solution



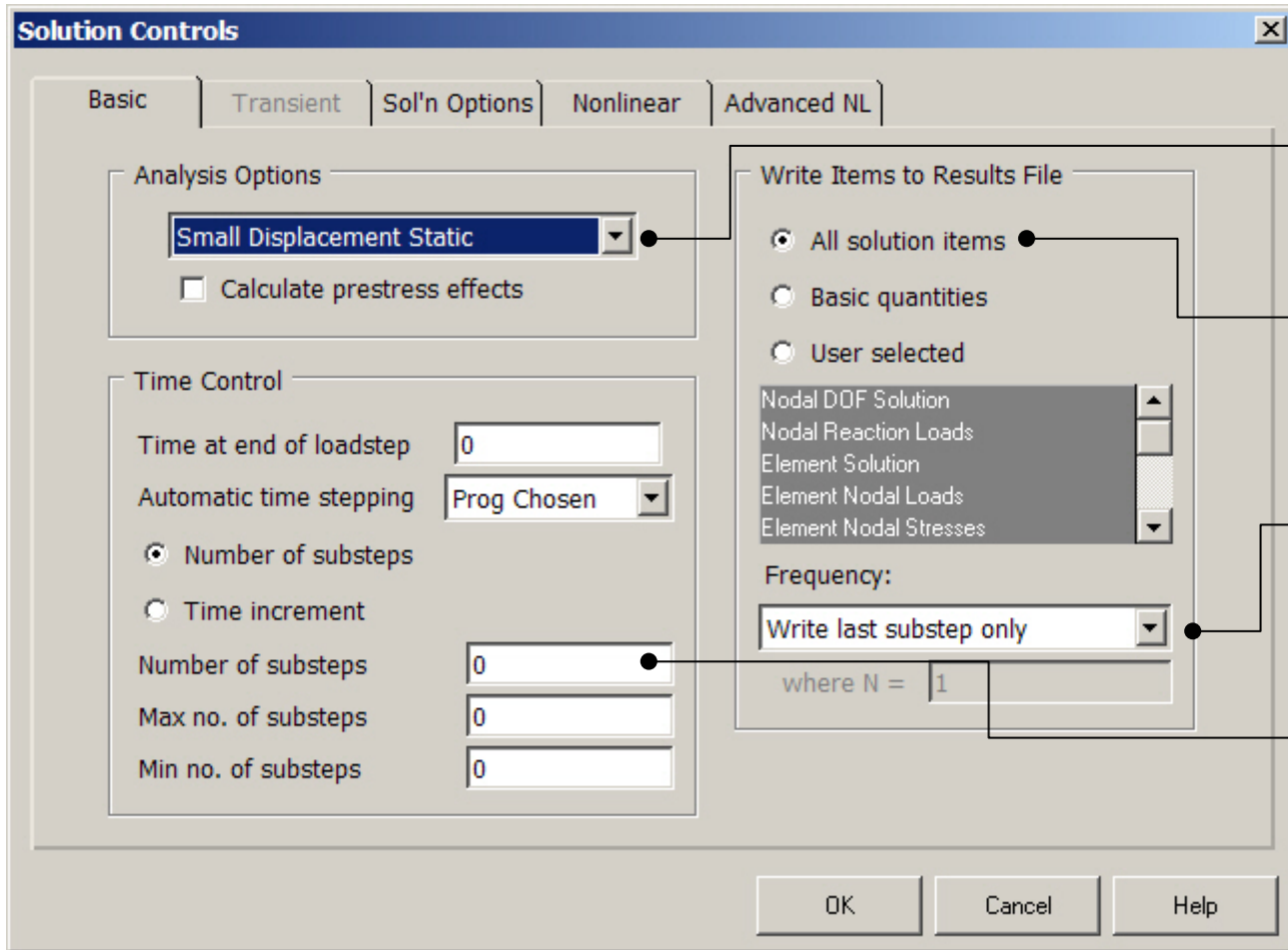
# Example – NL material models



# Example – Bilinear kinematic hardening



# Example – Solution Controls



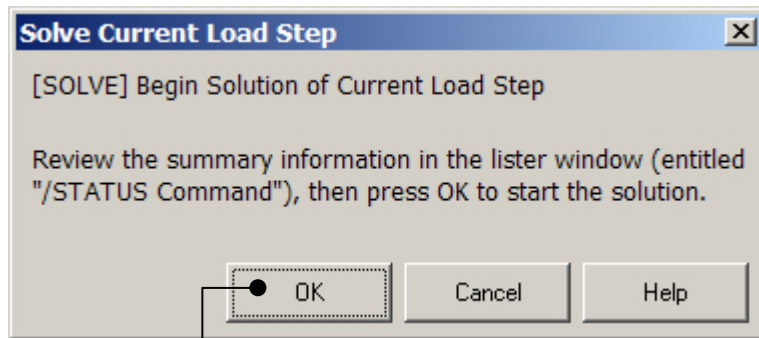
The screenshot shows the 'Solution Controls' dialog box with the 'Basic' tab selected. The 'Analysis Options' section has 'Small Displacement Static' selected in the dropdown menu. The 'Time Control' section has 'Time at end of loadstep' set to 0, 'Automatic time stepping' set to 'Prog Chosen', and 'Number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

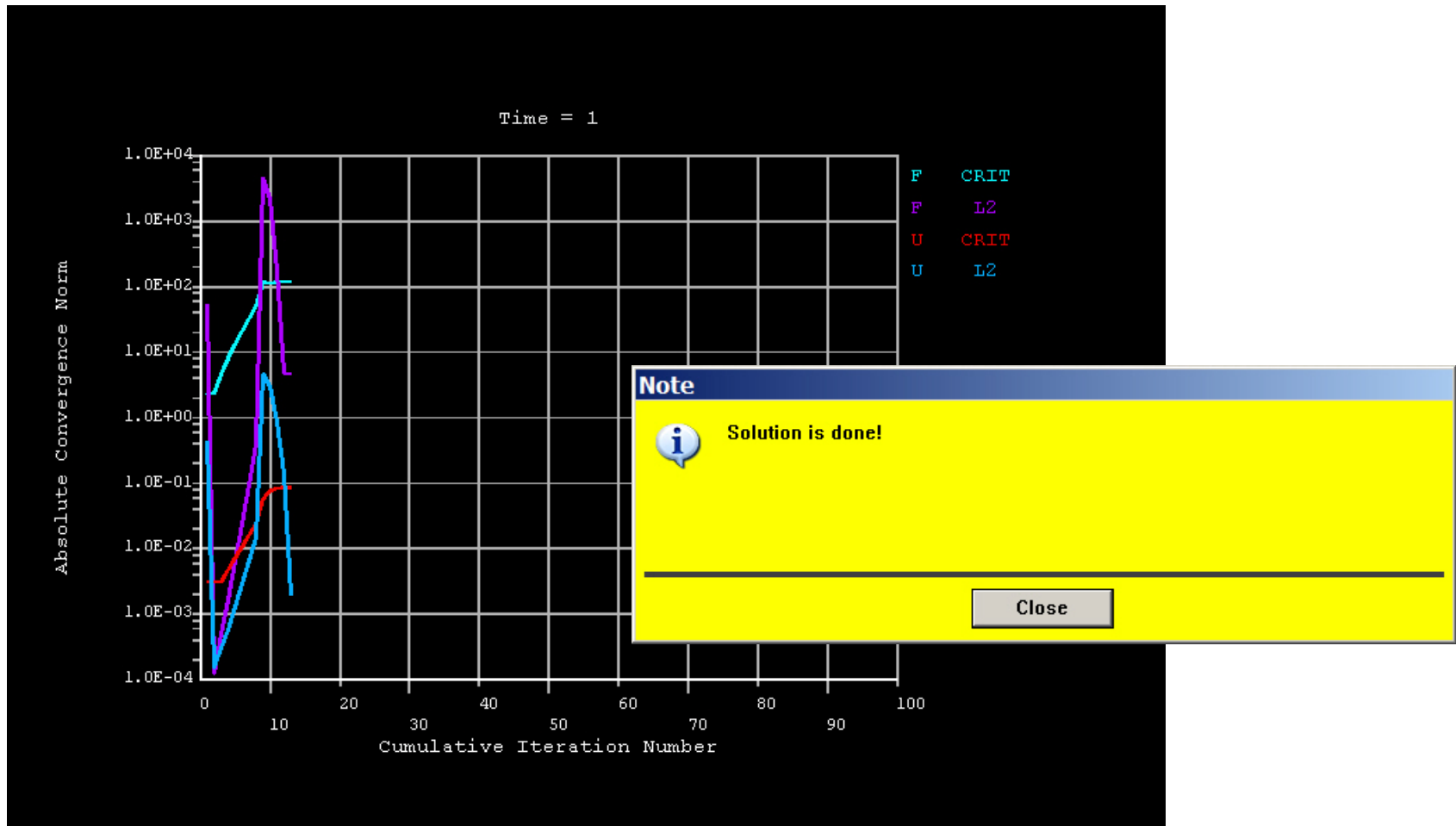
# Example - Solve

**Solution > Solve > Current LS**



Press OK

# Example - Convergence



# Example – NL material solution

