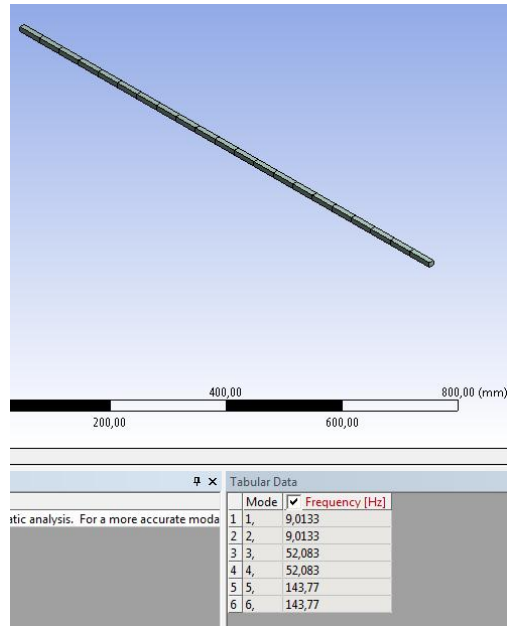


# Pre stress in a modal analysis Workbench 14.0



Aalborg Universitet esbjerg  
Søren Heide Lambertsen

File Edit View Units Tools Help

Show Vertices Wireframe Edge Coloring Thicken Annotations Show Mesh Show Coordinate Systems

Environment Inertial Loads Supports Conditions Direct FE

Outline

- Project
  - Model (B4)
    - Geometry
    - Coordinate Systems
    - Mesh
    - Static Structural (B5)
      - Analysis Settings
      - Fixed Support
      - Force
    - Solution (B6)
      - Solution Information

**B: Static Structural**  
 Force  
 Time: 1, s  
 23-02-2012 15:01

Force: 100, N  
 Components: 100, 0, 0, N

Details of 'Force'

Scope

Scoping Method	Geometry Selection
Geometry	1 Vertex

Definition

Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	100, N (ramped)
Y Component	0, N (ramped)
Z Component	0, N (ramped)
Suppressed	No

Graph

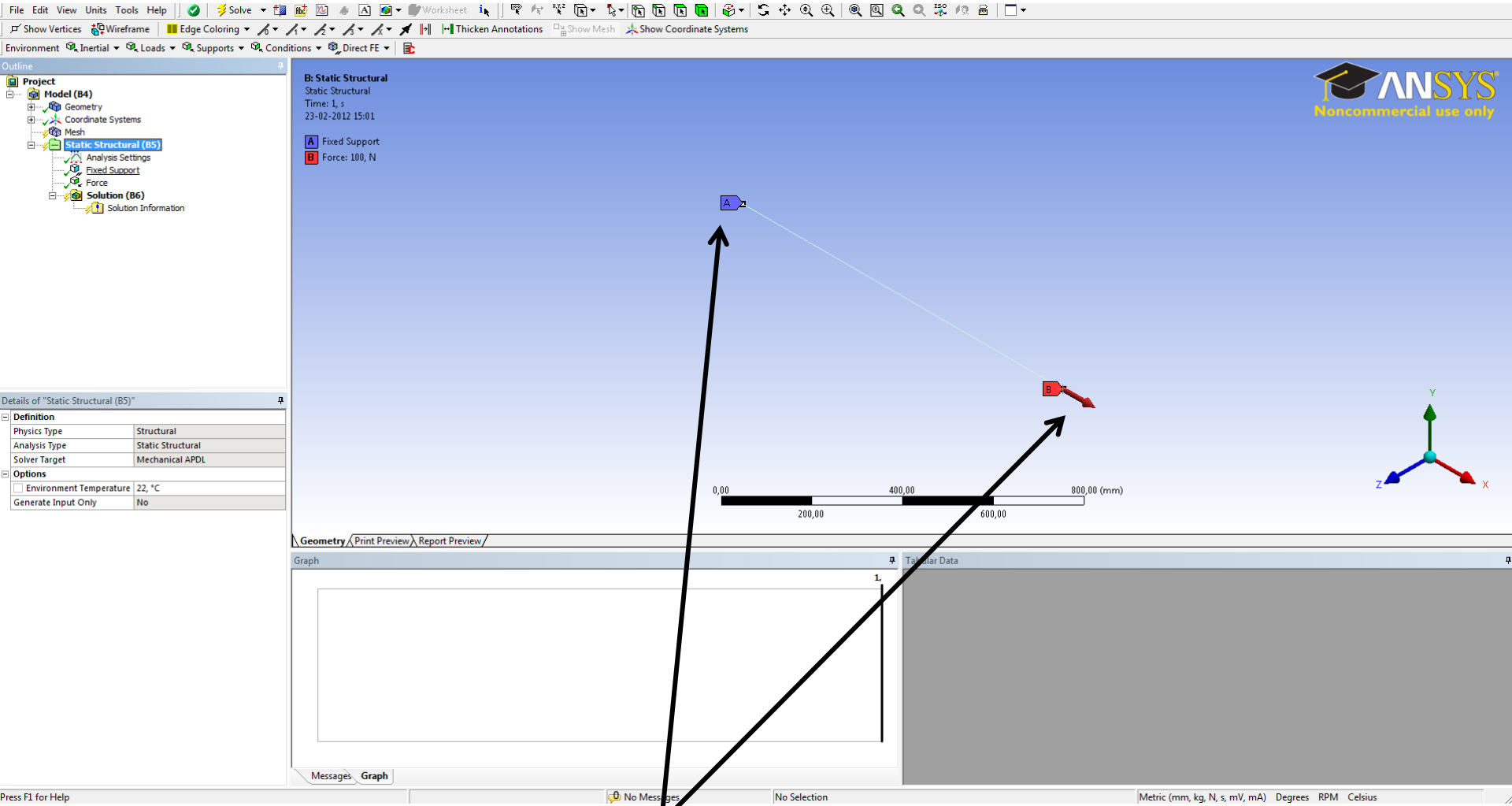
Tabular Data

Steps	Time [s]	X [N]	Y [N]	Z [N]
1	0	0	0	0
2	1	100	0	0

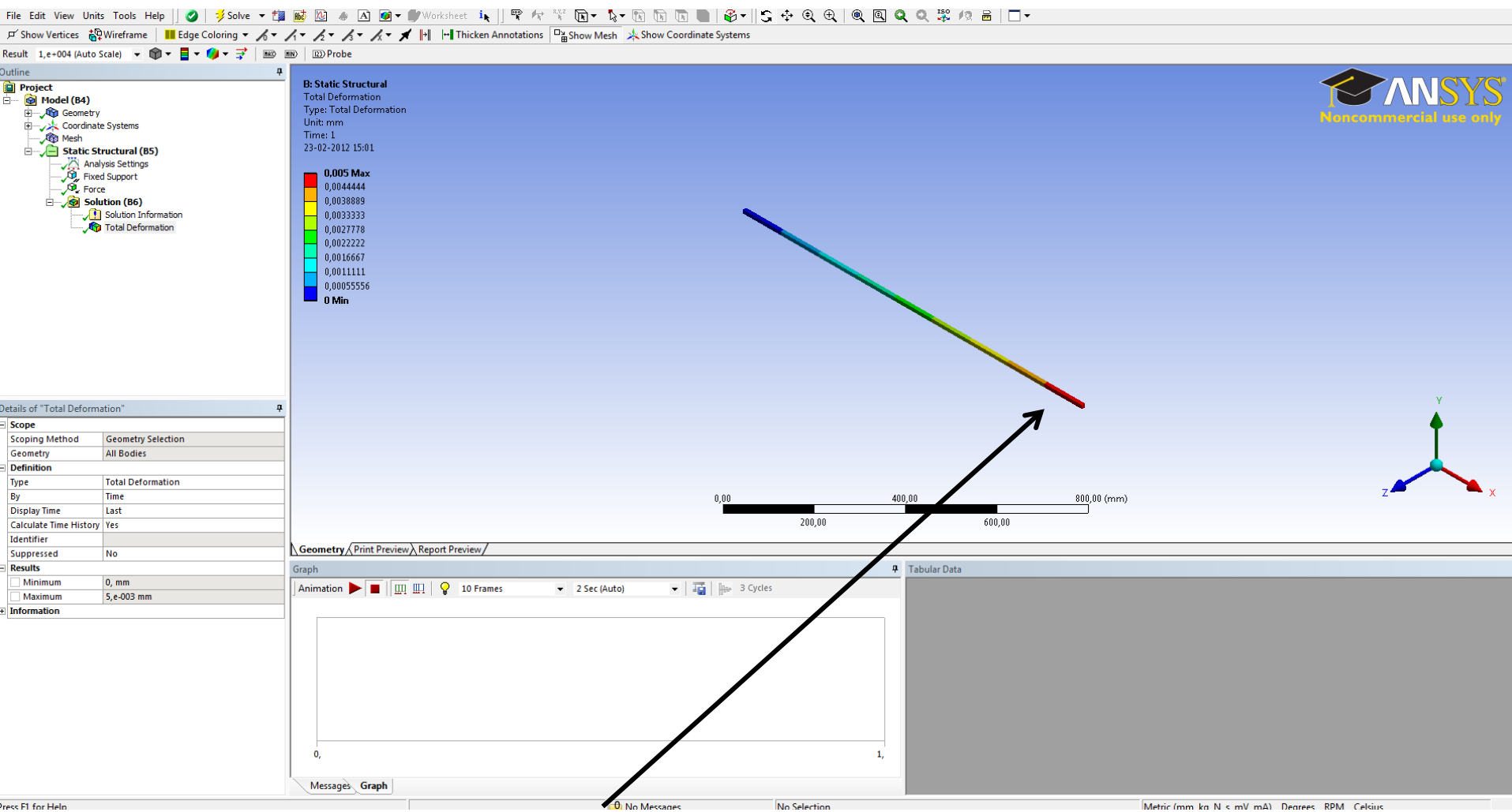
Press F1 for Help

No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees RPM Celsius

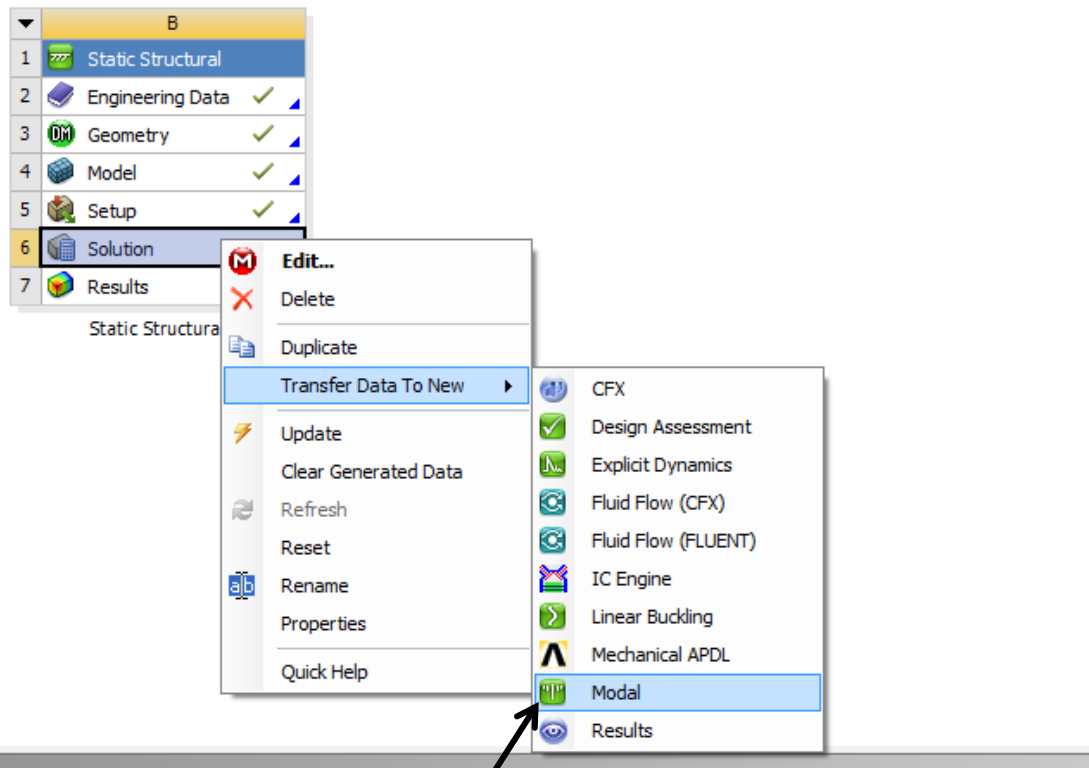
Start a static structural analysis and insert the boundary condition. In this ensamble a Force at 100 N is used.



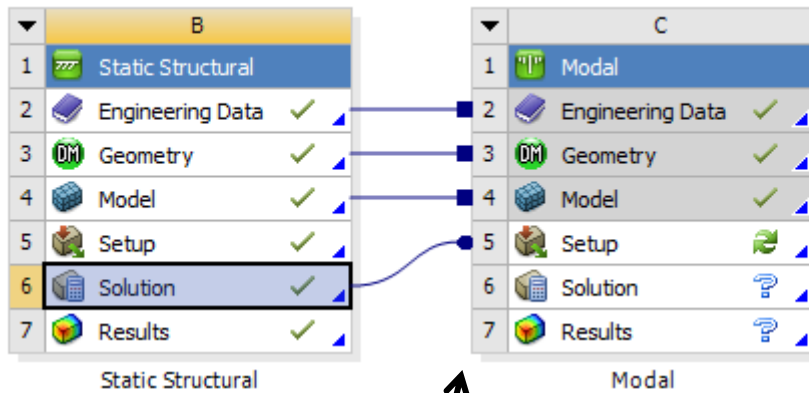
100 N force and a fixed support.



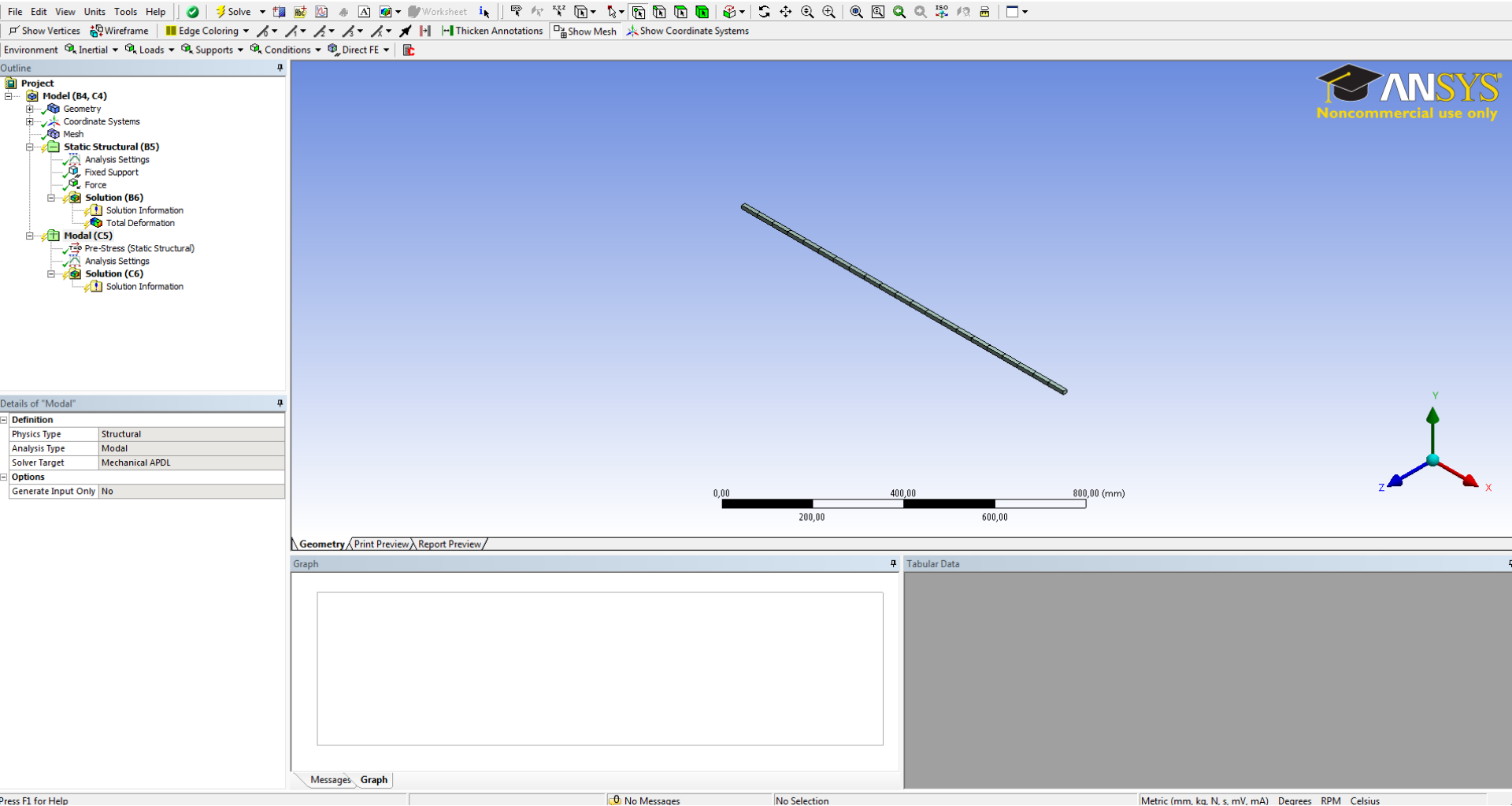
Then solve and plot the deformation and close the window.



Right click on the solution and click on Modal.



A modal analysis will start and use the data from the Static Structural. Double click on Setup at the modal analysis



Then press solve.

File Edit View Units Tools Help

Show Vertices Wireframe Edge Coloring Thicken Annotations Show Mesh Show Coordinate Systems

Solution Deformation Strain Stress Energy Probe Tools User Defined Result Campbell Diagram Coordinate Systems

ANSYS Noncommercial use only

Project Outline

- Model (B4, C4)
  - Geometry
  - Coordinate Systems
  - Mesh
  - Static Structural (B5)
    - Analysis Settings
    - Fixed Support
    - Force
  - Solution (B6)
    - Solution Information
    - Total Deformation
  - Modal (C5)
    - Pre-Stress (Static Structural)
    - Analysis Settings
    - Solution (C6)
      - Solution Information

Details of "Solution (C6)"

Adaptive Mesh Refinement

Max Refinement Loops 1,  
Refinement Depth 2

Information

Status Done

Geometry | Print Preview | Report Preview

Messages

Warning: You have performed a pre-stress modal analysis with large deflection effects turned off in the static analysis. For a more accurate modal analysis, you should turn on large deflection effects in the static analysis.

Tabular Data

Mode	Frequency [Hz]
1	9,0133
2	9,0133
3	52,083
4	52,083
5	143,77
6	143,77

0,00 200,00 400,00 600,00 800,00 (mm)

Press F1 for Help

1 Message No Selection Metric (mm, ka, N, s, mV, mA) Degrees RPM Celsius

Under solution the 6 first frequencies.