Cantilever Beam Modal Analysis - Numerical Solution

Author: John Singleton, Cornell University

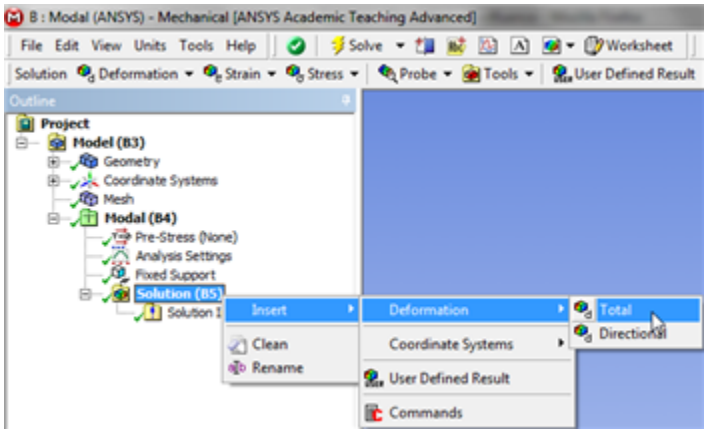
Problem Specification
1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Numerical Solution
6. Numerical Results
7. Verification & Validation
Exercises
Comments

Numerical Solution

Specify Results (Deformation)

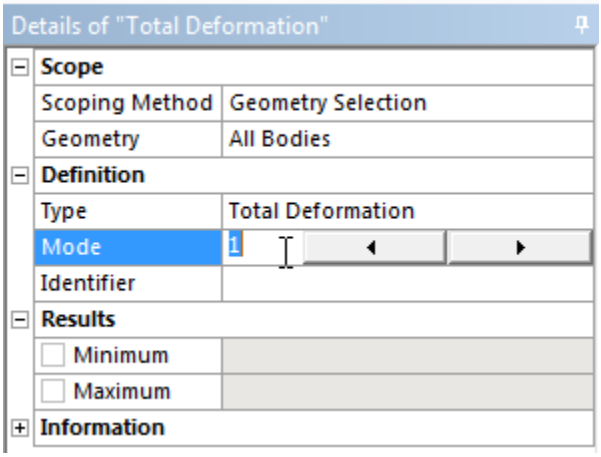
Here, we will tell ANSYS to find the deformation for the first six modes. Then, we will be able to see the shapes of the six modes. Additionally, we will be able to watch nice animations of the six modes.

In order to request the deformation results ***(right click) Solution > Insert > Deformation > Total*** as shown below.

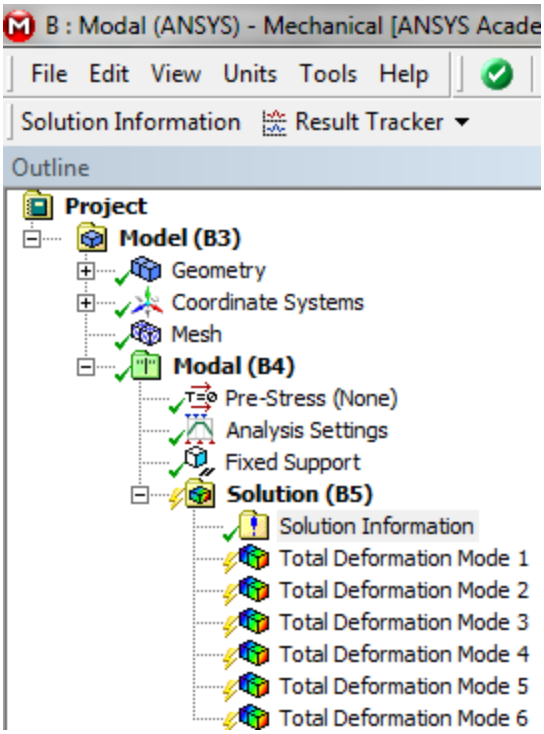


Click Here for Higher Resolution

Then, rename "Total Deformation" to "Total Deformation Mode 1". In order to do so ***(right click) Total Deformation > Rename***. Next, set ***Mode*** to 1 as shown in the image below.



Repeat, this process for the other 5 modes. Make sure that you set ***Mode*** to the respective mode number. At this point, your ***Outline*** should look the same as the following image.



Run Calculation

In order to run the simulation and calculate the specified outputs, click the ***Solve*** button,

.

Save

Go to Step 6: Numerical Results

Go to all ANSYS Learning Modules