# Cantilever Beam Modal Analysis - Physics Setup

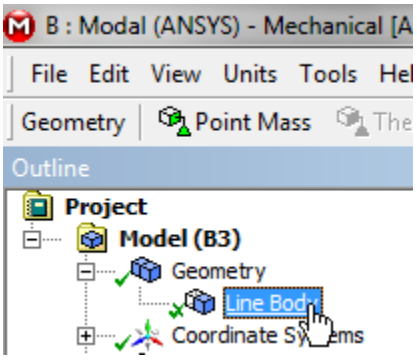
Author: John Singleton, Cornell University

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

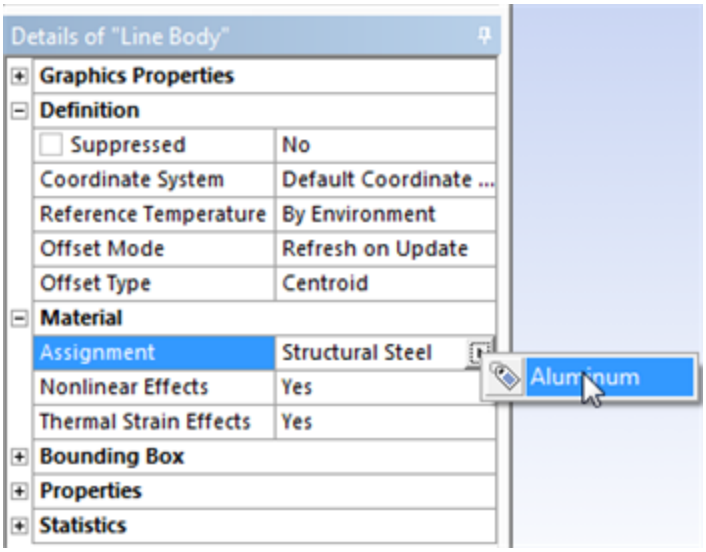
## Physics Setup

**Material Assignment**

At this point, we will tell ANSYS to assign the Aluminum material properties that we specified earlier to the geometry. First, ***(expand) Geometry*** then ***(click) Line Body***, as shown below.

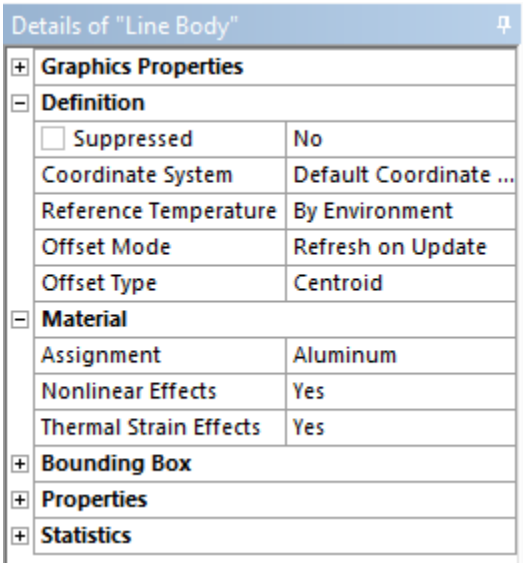



Then, ***(expand) Material*** in the "Details of Line Body" table and set ***Assignment*** to Aluminum, as shown below.



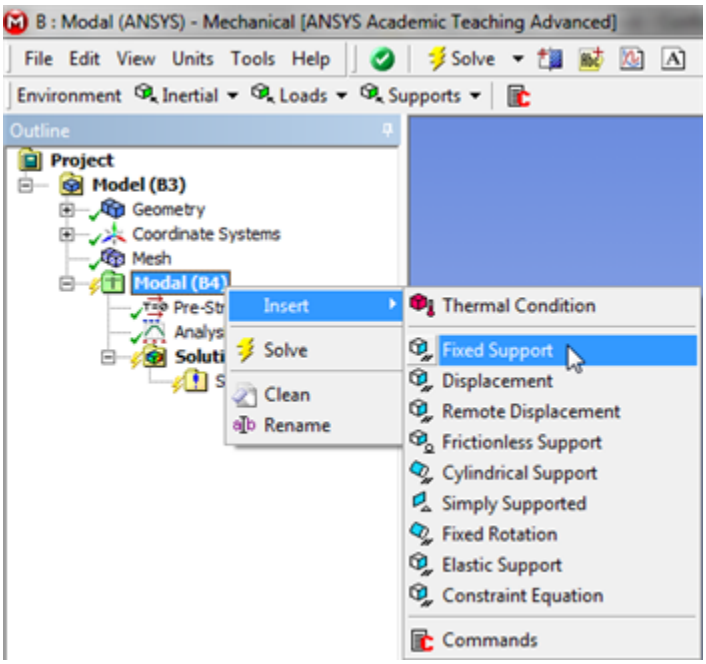

Click Here for Higher Resolution

At this point your "Details of Line Body" table, should look comparable to the following image.




### Fixed Support

First, ***(right click) Modal > Insert > Fixed Support***, as shown below.

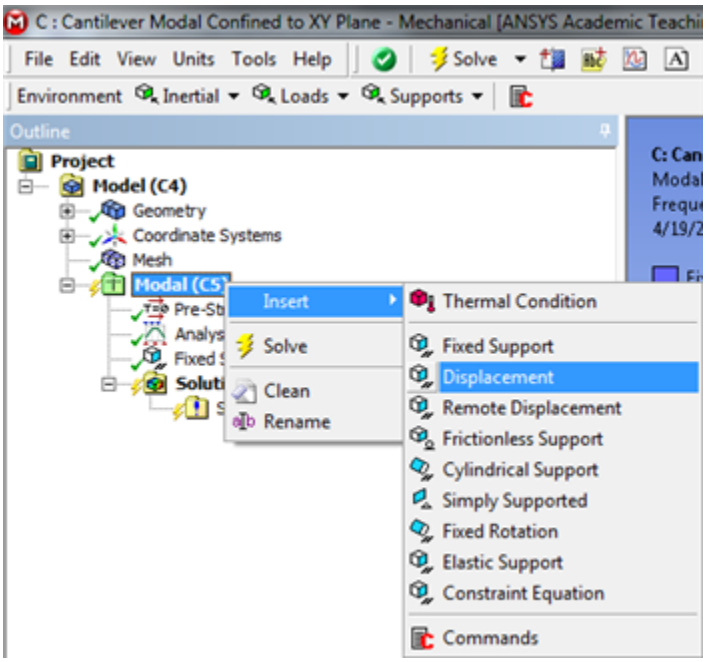



Next, click the ***vertex selection filter*** button, . Then, click on the left end of the beam and apply it as the ***Geometry*** in the "Details of Fixed Support" table.

### Constrain Beam to XY Plane

In this section the beam's motion will be constricted to the xy plane.

First, ***(right click) Modal > Insert > Displacement***, as shown below.




Click Here for Higher Resolution

Next, click the ***edge selection filter*** button, . Then, click on the geometry and apply it as the ***Geometry*** in the "Details of Displacement" table. Lastly, set ***Z Component*** to 0, as shown below.

Details of "Displacement"

| | |
|---|---|
| **Scope** | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Edge |
| **Definition** | |
| Type | Displacement |
| Define By | Components |
| Coordinate System | Global Coordinate System |
| X Component | Free |
| Y Component | Free |
| Z Component | 0 |
| Suppressed | No |

## Save

**Go to Step 5: Numerical Solution**

Go to all ANSYS Learning Modules