

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](#)

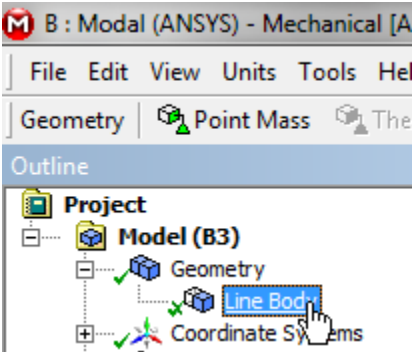
[Exercises](#)

[Comments](#)

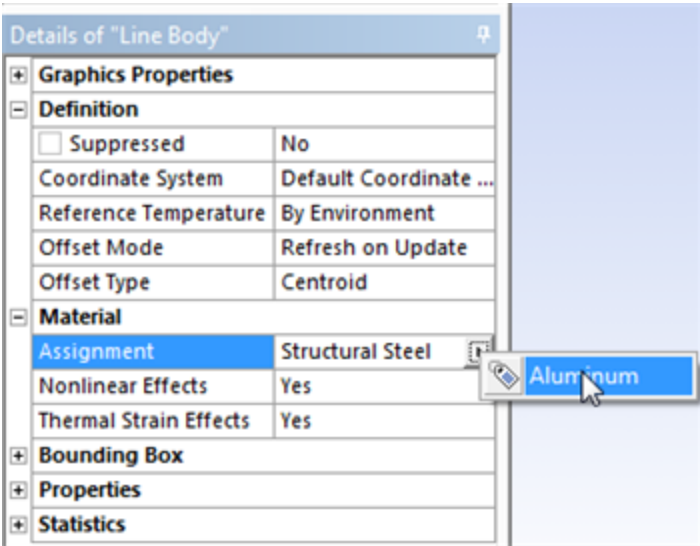
Physics Setup

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.

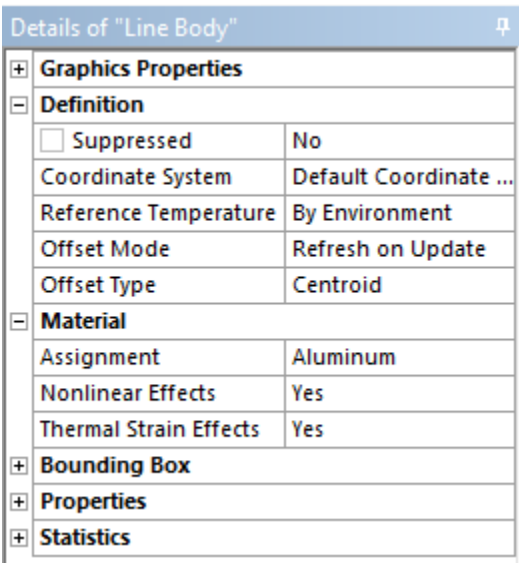


Then, *(expand) Material* in the "Details of Line Body" table and set *Assignment* to Aluminum, as shown below.



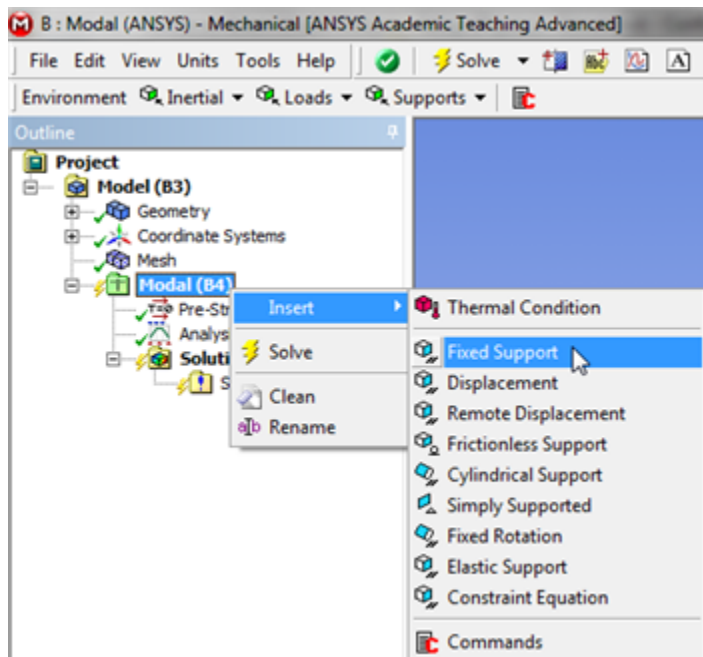
[Click Here for Higher Resolution](#)


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.

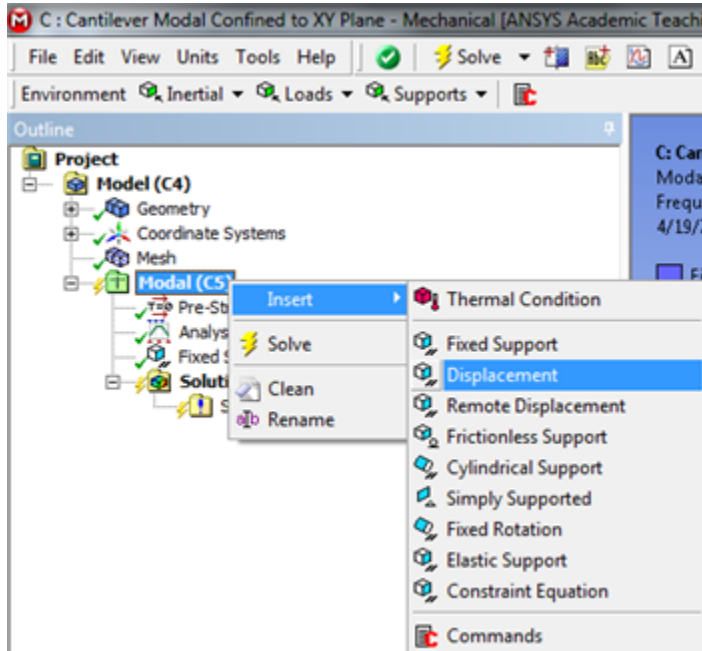


Next, click the *vertex selection filter* button, . Then, click on the left end of the beam and apply it as the *Geometry* in the "Details of Fixed Support" table.


Constrain Beam to XY Plane

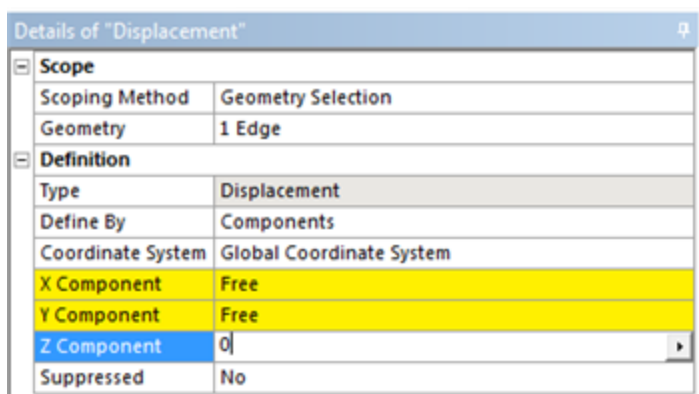
In this section the beam's motion will be constricted to the xy plane.

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.



Save

[Go to Step 5: Numerical Solution](#)

[Go to all ANSYS Learning Modules](#)