

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](#)

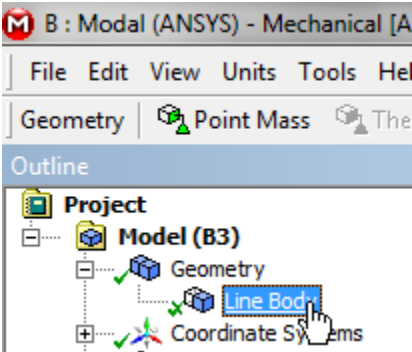
[Exercises](#)

[Comments](#)

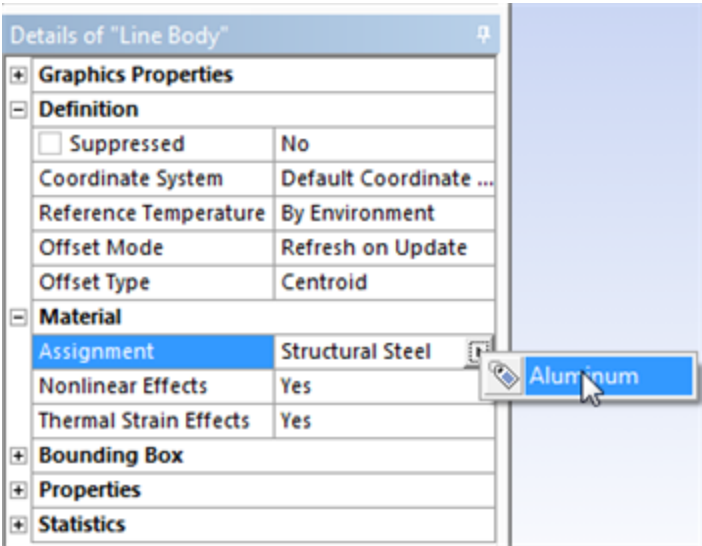
Physics Setup

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.

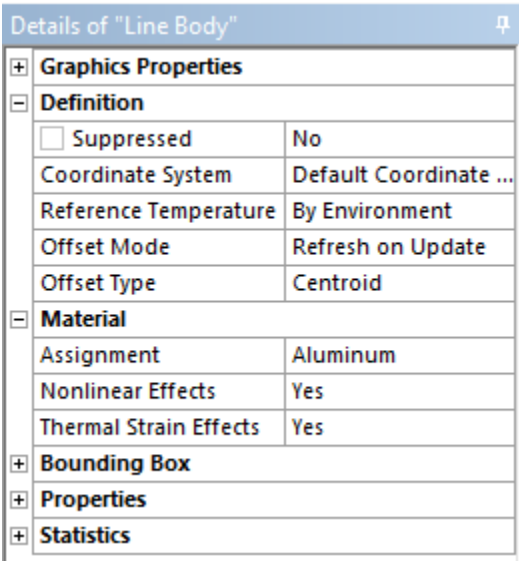


Then, *(expand) Material* in the "Details of Line Body" table and set *Assignment* to Aluminum, as shown below.



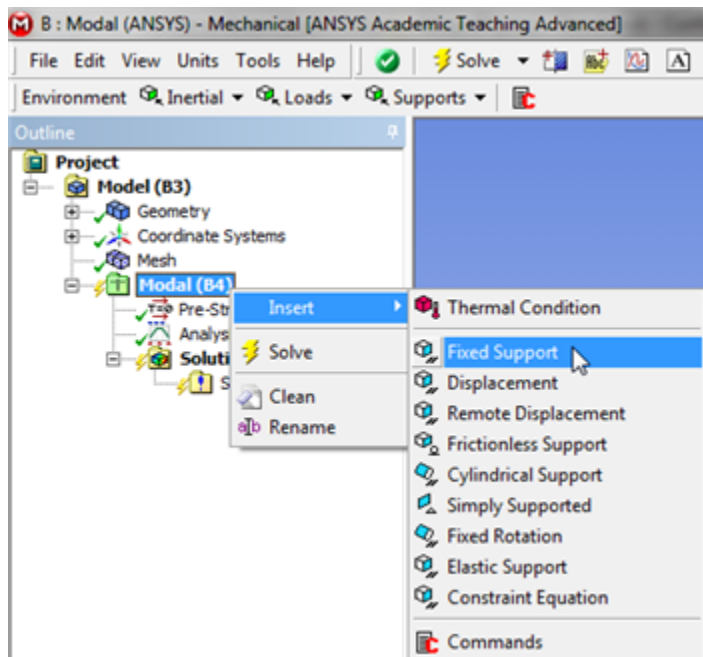
[Click Here for Higher Resolution](#)

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.

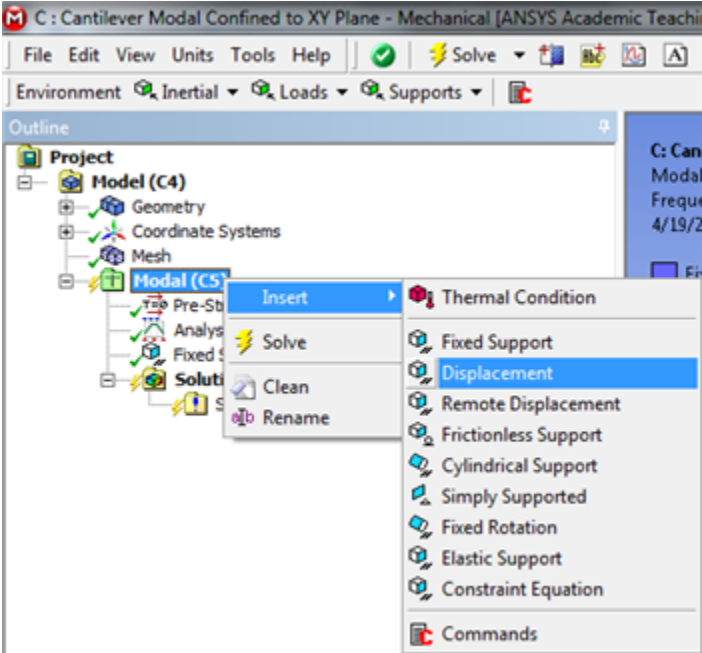


Next, click the *vertex selection filter* button, . Then, click on the left end of the beam and apply it as the *Geometry* in the "Details of Fixed Support" table.

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](#)