Cantilever Beam - Physics Setup (OLD)

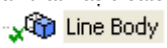
Author: John Singleton, Cornell University

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

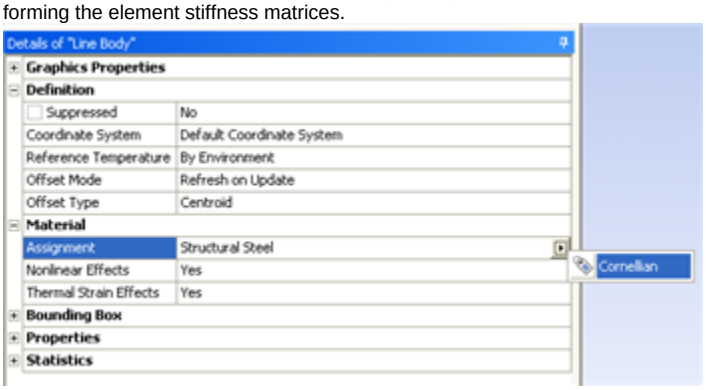
Physics Setup

Assign Material Properties

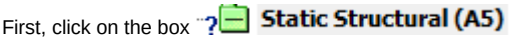
The material Cornellian that was created earlier needs to be applied to the beam. In order to do so, expand ***Geometry***, 

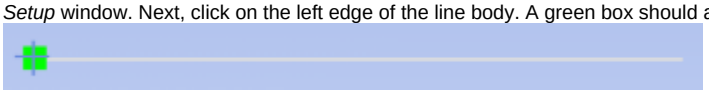
. Next, click once on ***Line Body***, 

, which will appear underneath Geometry. Then expand ***Material*** which is located under *Details of Line Body*. Then click on the arrow on the far right and change the specified material to Cornellian as shown below. Now ANSYS will use the correct Young's Modulus while forming the element stiffness matrices.

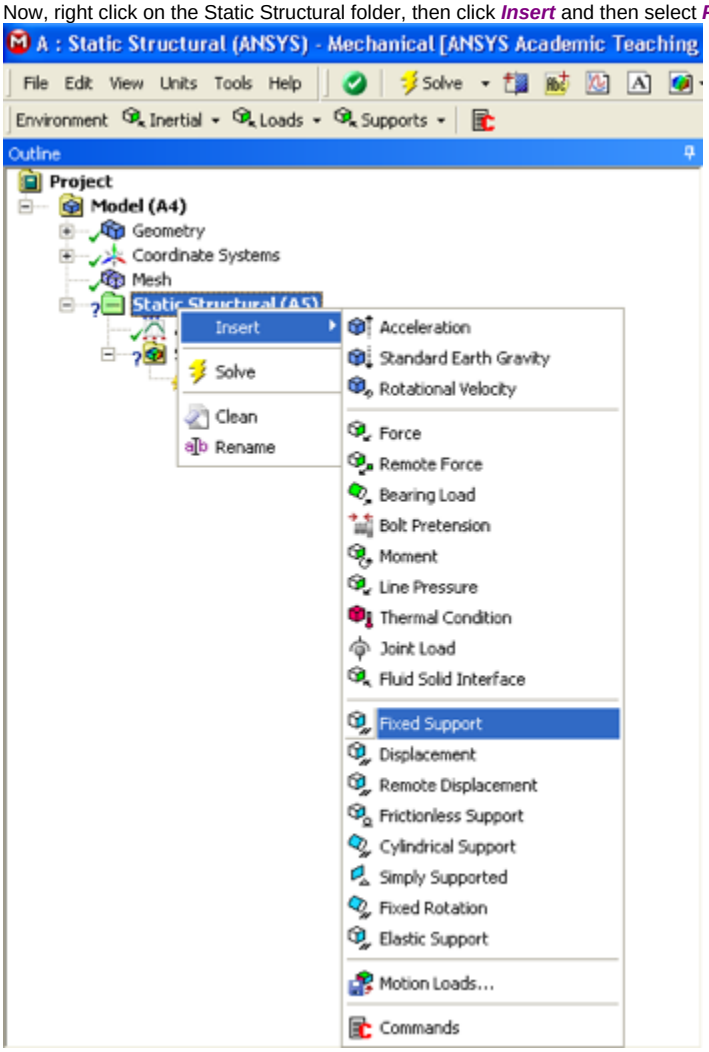


Fix The Left Side of The Beam

First, click on the box  in the *Outline Window*. Next, click on the vertex pointer option,  , which is on the top of the *Setup* window. Next, click on the left edge of the line body. A green box should appear on the left end of the line body as seen below.



Now, right click on the Static Structural folder, then click ***Insert*** and then select ***Fixed Support*** as shown in the image below.

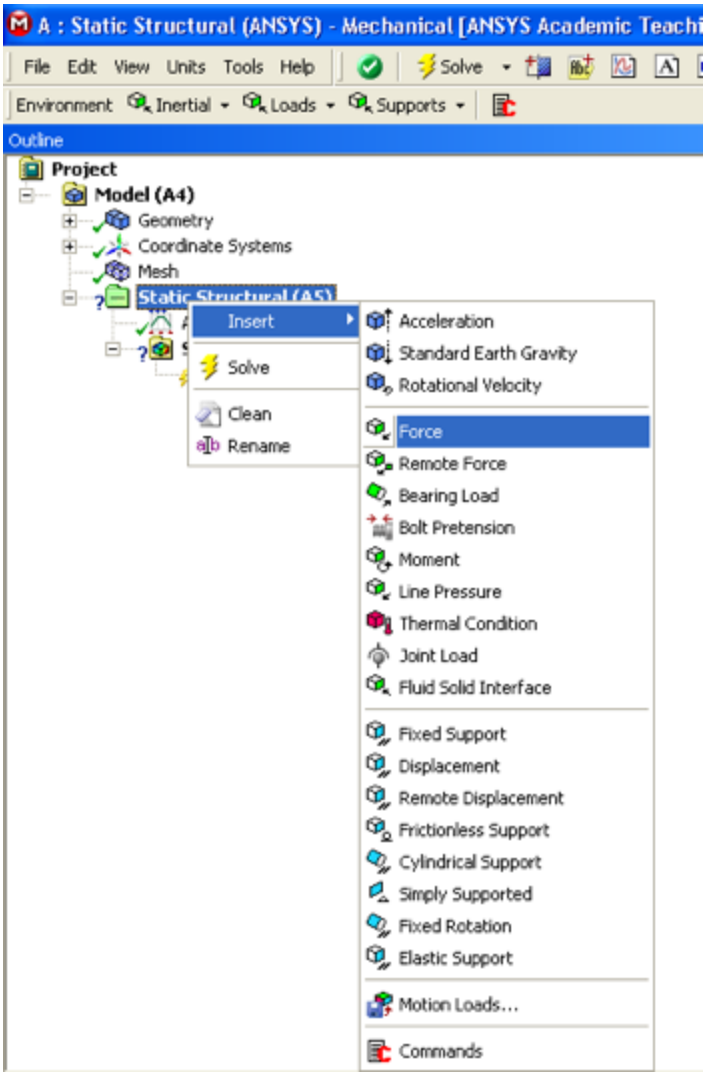


This will set the *x* and *y* displacements as well as the slope to zero for the node at the left end of the neutral axis.

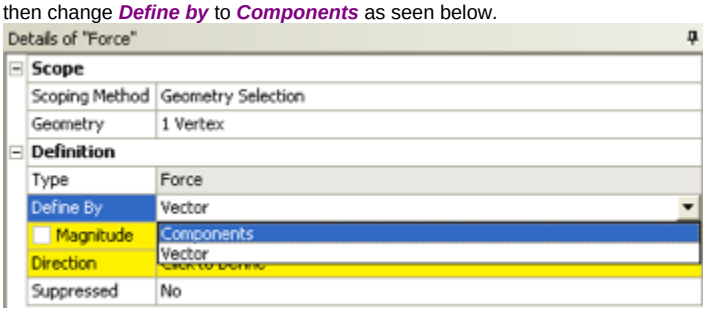
Apply a Point Force to The Right Side of The Beam

Once again, click on the vertex pointer option,  , but now click the right edge of the line body. A green box should now appear on the right side of the line body as seen below.

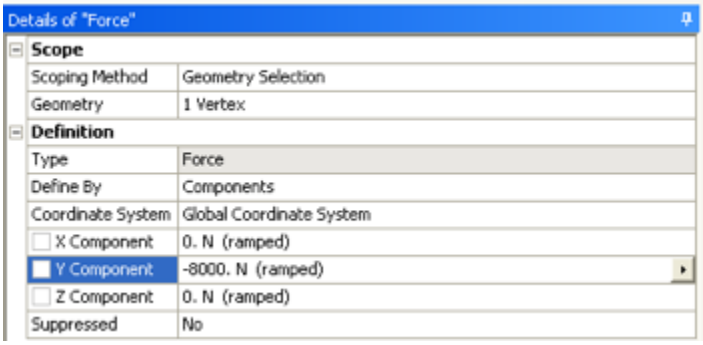
Right click on the Static Structural folder again, then click ***Insert*** and this time select ***Force*** as shown below.



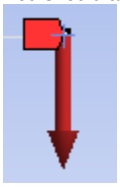
At this point, there should be a *Details of "Force"* window in the lower left corner of the Setup window. Expand ***Definition*** if it is not already expanded and then change ***Define by*** to ***Components*** as seen below.



Now, click on the box to the right of Y Component without clicking the Y component button and change the force to `-8000N`. That is, you should NOT check the box to the left of Y Component. Your *Details of "Force"* window should now look very similar to the following image.



You should also see the following downward facing red arrow on the right side of the line body.



The fixed end and the point force have now been applied. Leave the Setup window open for the next step. Save the project.

Go to Step 5: Numerical Solution

Go to all ANSYS Learning Modules