

Cantilever Beam - Mesh (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](#)

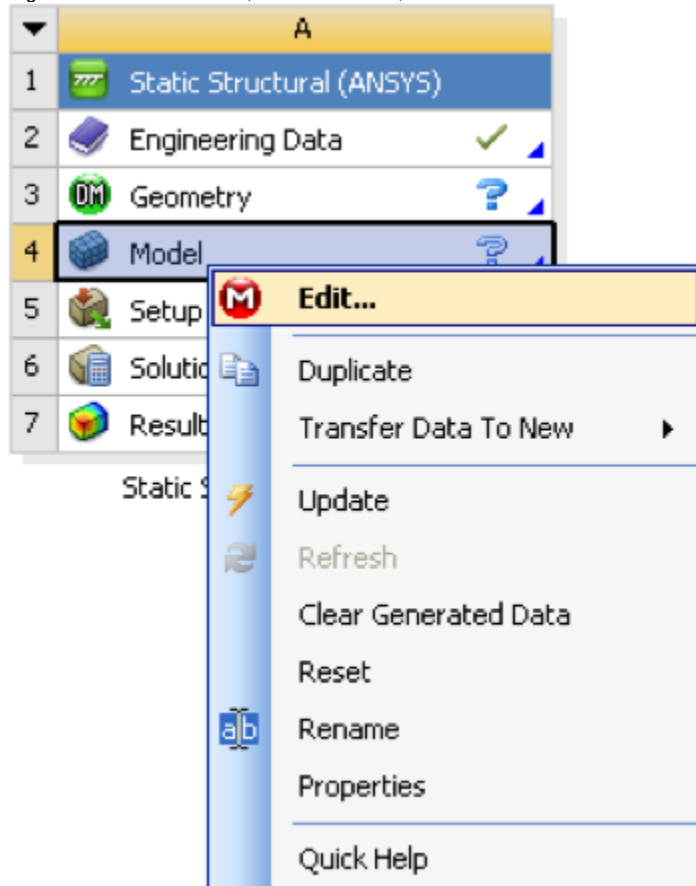
[Exercises](#)


[Comments](#)

Mesh



Open the Model

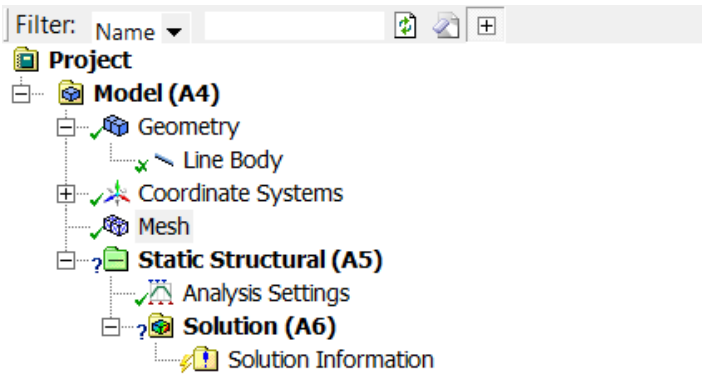
Right click on model button,  Model, in the Workbench window then click on *Edit...* as shown below.



Expand "Model (A4)",  Model (A4), if it is not already expanded.

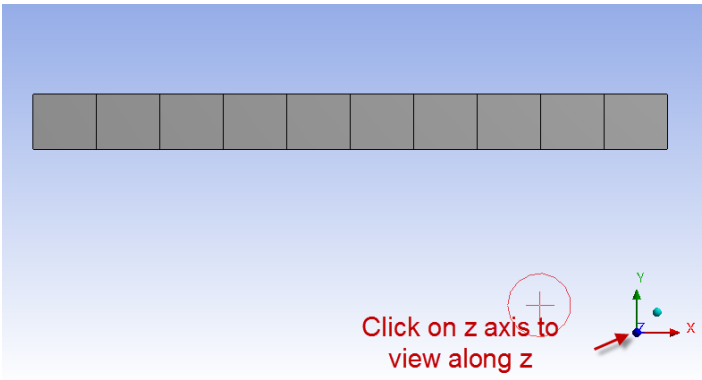
Specify the Element Size

Begin by clicking once on mesh,  **Mesh** . Next, expand sizing under *Details of "Mesh"* if it is not already expanded. To create ten elements along the beam, set the element size to $4\text{m}/10 = 0.4\text{m}$. Then, click on **Update**,  **Update** .



Details of "Mesh"	
Defaults	
Physics Preference	Mechanical
<input type="checkbox"/> Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
<input type="checkbox"/> Element Size	0.40 m
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	4.0 m
Inflation	

At this point you should see a similar image to the one below. The mesh is composed of ten elements.



The mesh has now been set.



Note

In the above mesh view, ANSYS is wrapping the cross-section around the line elements. To view the line elements and the corresponding nodes, click on **View** and uncheck **Thick Shells and Beams**. ANSYS will calculate the displacements and slopes at the nodes shown in this view.



[click here for full view](#)

[Go to Step 4: Physics Setup](#)

[Go to all ANSYS Learning Modules](#)