

Cantilever Beam Modal Analysis - Mesh

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](#)

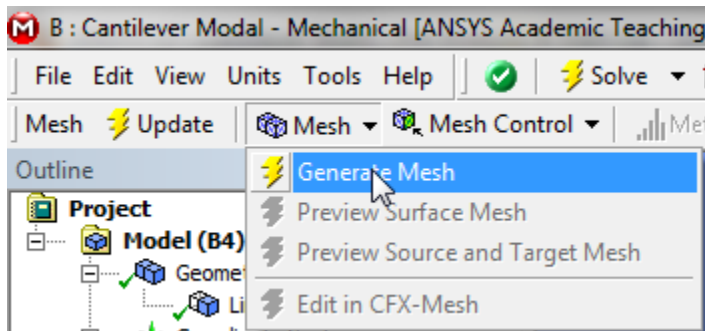
Mesh

Launch Mechanical

(double click) **Model**,  **Model**, in the "Cantilever Modal" project.

Generate Default Mesh

First, (click) **Mesh** in the tree outline. Next, (click) **Mesh > Generate Mesh** as shown below.

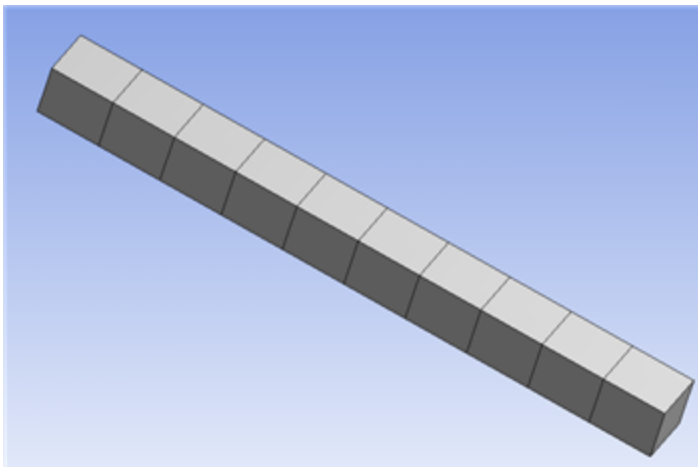


Size Mesh

In this section we will size the mesh, such that it has ten uniform elements. In order to size the mesh, first expand **Sizing** located within the **Details of "Mesh"** table. Next, set **Element Size** to 0.40 m, as shown below.

Details of "Mesh"		
[-] Defaults		
Physics Preference	Mechanical	
Relevance	0	
[-] Sizing		
Use Advanced Size Function	Off	
Relevance Center	Coarse	
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		
[+] Advanced		
[+] Pinch		
[+] Statistics		

Now, (click) **Mesh > Generate Mesh** in order to generate the new mesh. You should obtain the mesh, that is shown in the following image.



[Click Here for Higher Resolution](#)

Note that in this simulation we are working with beam elements, which are simply line segments. As a visualization tool ANSYS displays a beam with width and height. In order to display the actual mesh (click) **View > (deselect) Thick Shells and Beams**. You will then see the mesh displayed in its native form.



[Click Here for Higher Resolution](#)

Save

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

[Go to all ANSYS Learning Modules](#)