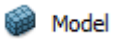
Cantilever Beam Modal Analysis - Mesh

Author: John Singleton, Cornell University

Mesh

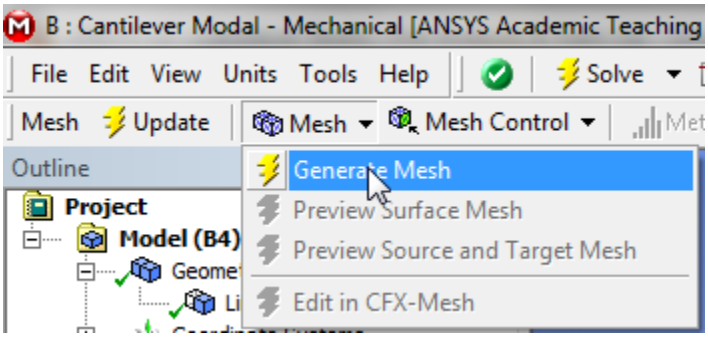
Launch Mechanical

(double click) Model, 

, in the "Cantilever Modal" project.

Generate Default Mesh

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

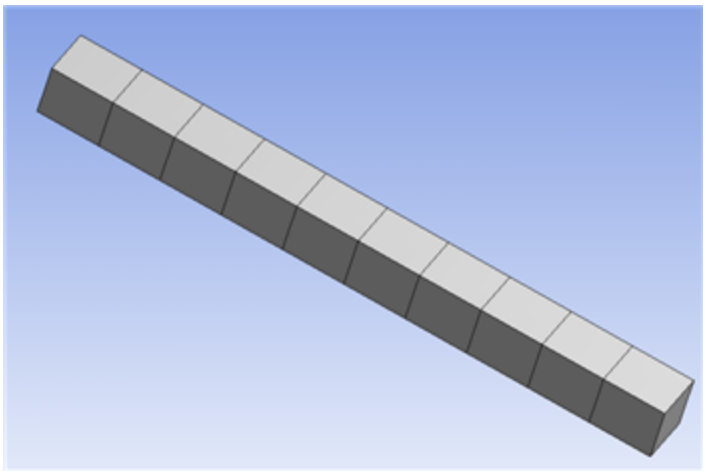


Size Mesh

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