

# 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](#)

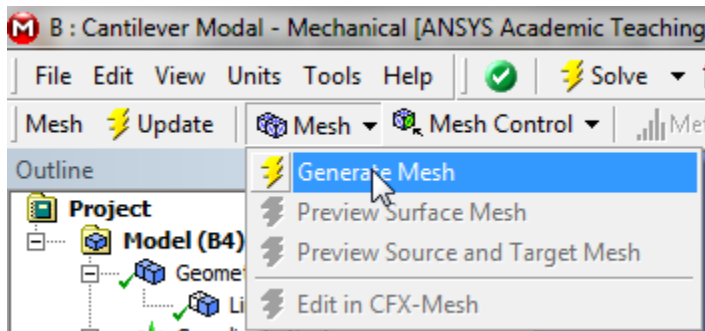
## Mesh

### Launch Mechanical

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

### Generate Default Mesh

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

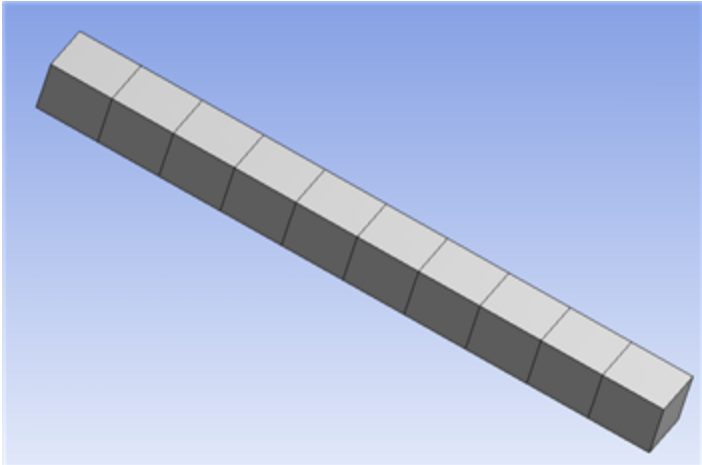


### Size Mesh

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](#)