

Cantilever Beam Modal Analysis - Geometry

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

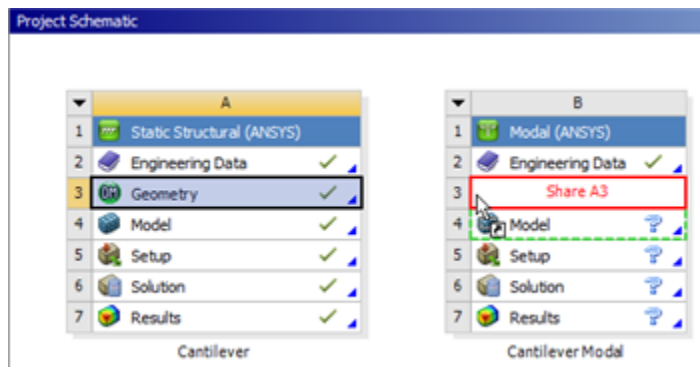
Geometry



For users of ANSYS 15.0, please check [this link](#) for procedures for turning on the Auto Constraint feature before creating sketches in DesignModeler.

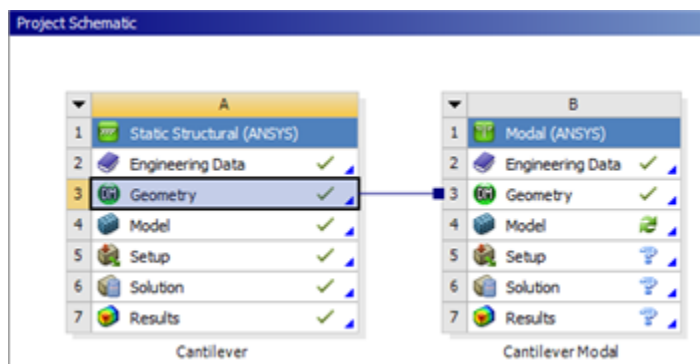
Attach Geometry from Cantilever to Cantilever Modal

The geometry for the "Cantilever Beam Modal Analysis" tutorial is the same as the geometry for the "Cantilever Beam" tutorial. Instead of recreating the geometry, we will simply attach the geometry from the Static Structural Analysis System (Cantilever) to the Modal Analysis System (Cantilever Modal). In order to attach the geometry, (*left click*) **Geometry** in the "Cantilever" project and drag it to **Geometry** in the "Cantilever Modal" project, as shown below.



[Higher Resolution Image](#)

Then release the left mouse button. You should now see that the geometries are shared as shown in the following image.



[Higher Resolution Image](#)

Save

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)