

Modal Analysis of a Composite Monocoque - Geometry

Author: Jingsi Wu, 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

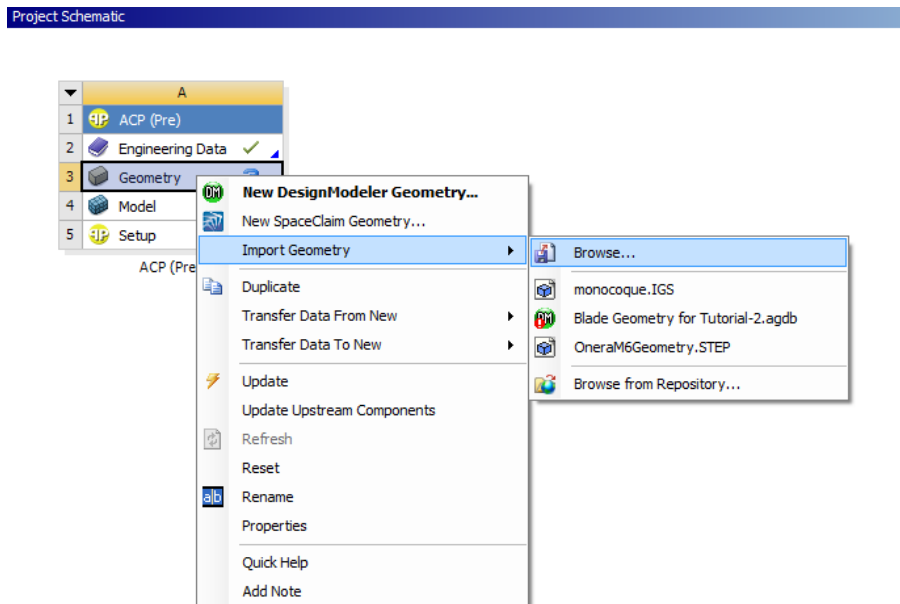
Exercises

Comments

Geometry

Import the Geometry

Please download the .igs file for the geometry here: [monocoque.IGS](#). To import the geometry, right-click **Geometry**, mouse-over **Import Geometry** and select **Browse**. Then, navigate to the location where you saved the monocoque.igs file and click Open. A green checkmark should now appear next to Geometry.



Build the Geometry

Double-click **Geometry** to bring up Design Modeler. On the top toolbar, select **Units** and then choose **inch**.

This particular geometry contains solids, surfaces and line bodies. In default setting, ANSYS does not build the line bodies, but we can change that easily. Under Detail of Import1, change Line Bodies to **Yes** as shown below:

Details View

Details of Import1

Import	Import1
Source	C:\Users\en-fsae\Desktop\monocoque.IGS
Base Plane	XYPlane
Operation	Add Material
Solid Bodies	Yes
Surface Bodies	Yes
Line Bodies	Yes
Simplify Geometry	No
Simplify Topology	No
Tolerance	Normal
Replace Missing Geometry	No
Stitch Surfaces	Yes

Now Click **Generate** to build the geometry.

This will make sure line bodies will be built along with the surfaces and solids.

Prepare the geometry for meshing

The following video shows how to prepare the geometry for composite analysis with ANSYS.

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)