

Modal Analysis of a Wing - Geometry

Author: Benjamin Mullen, 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](#)

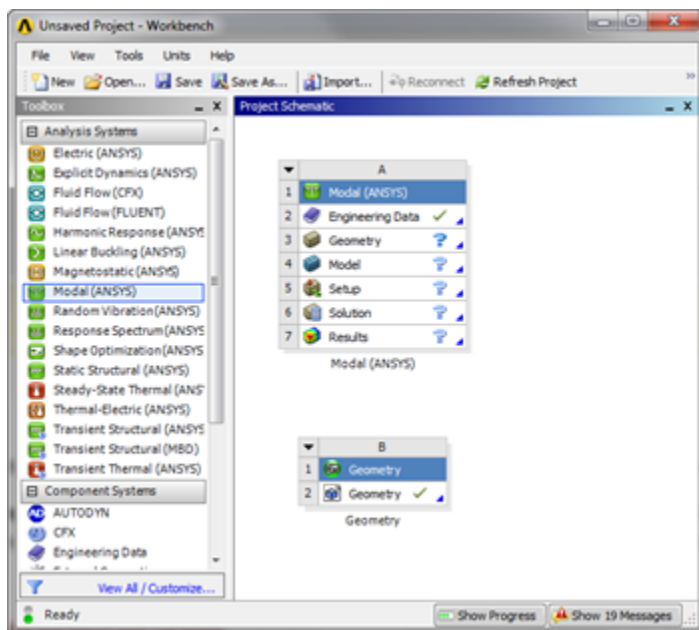
Geometry





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

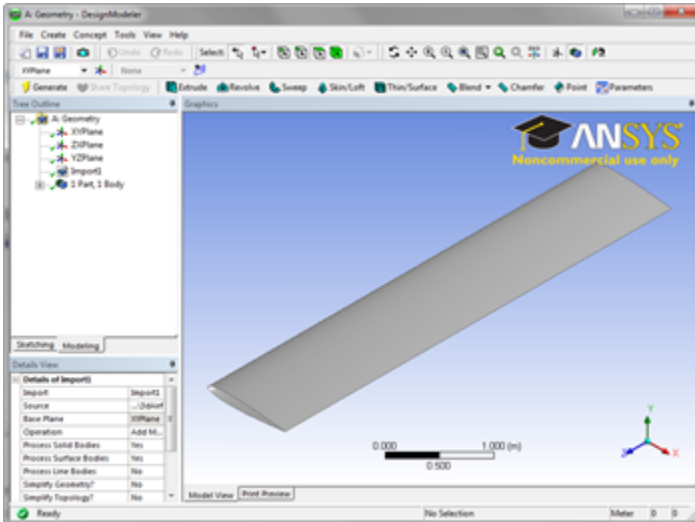
For this problem, we are going to import the geometry into ANSYS from a CAD package. Download the part file [here](#). Save the file somewhere convenient, like your desktop or your working directory.

To open the file in ANSYS, go to **File > Import**. Browse to the geometry location on your computer. If you do not see the file, make sure you are browsing for **geometry files** (the pull down menu at the bottom right of the browsing window for computers running Windows 7). Select the Geometry and click **Open**. This will import your geometry into ANSYS. Your project window should now include the main project, and the newly imported geometry (see below).



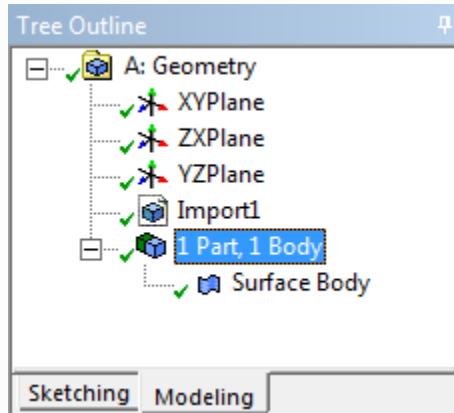
Now that the geometry has been imported, let's open the file and make sure everything is in order! Double click  **Geometry**. This will open the design modeler. When you are prompted, select **Meter** as your standard unit of measurement. The first thing you should notice is that the geometry is not

there, so click  **Generate** to generate the geometry. When the geometry finally generates, you should see the screen below.





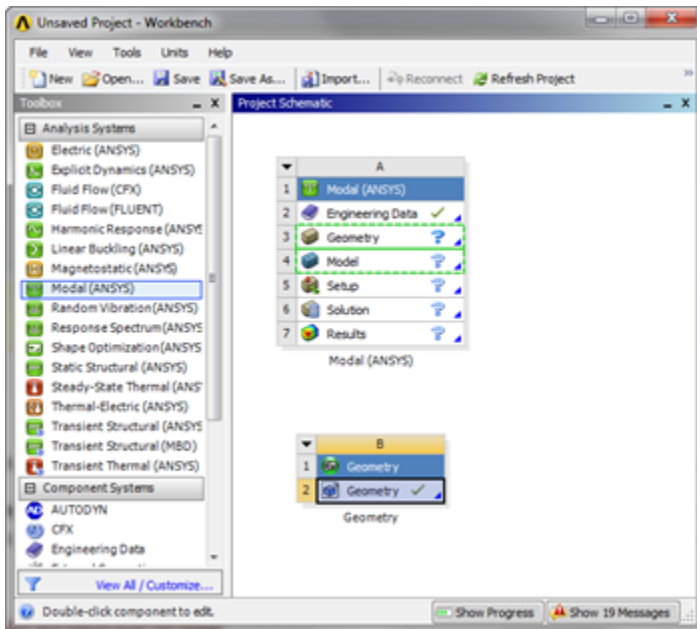
Importing Anomaly


Look at the *Outline* window in the design modeler. It should look like this:

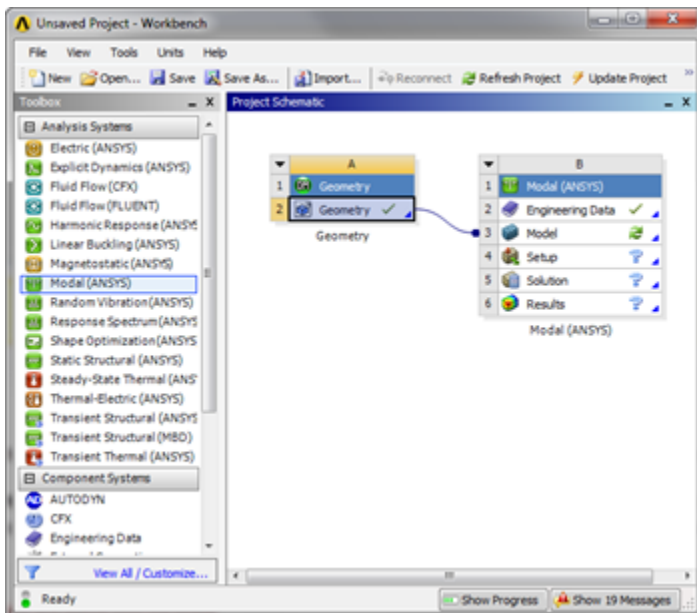


Specifically, there should be a **1 Part, 1 Body** expandable tab. Expand the tab and the only contents should be **Surface Body**. In certain versions of ANSYS, it has been reported that superfluous line bodies are being imported as well. If you see the line bodies in this window, right click each of the line bodies and select **Supress**. The line bodies will cause errors when we are meshing the body.

Once we are satisfied with our geometry, we can close the design modeler. Now, we should be looking at the *Project* window. To connect the geometry to the project, click and drag   **Geometry**. As soon as you drag the box, ANSYS will highlight the geometry and model boxes in the main project.



Drag and drop the geometry box onto  Model



The geometry has been connected the project and we are ready for the next step.

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)