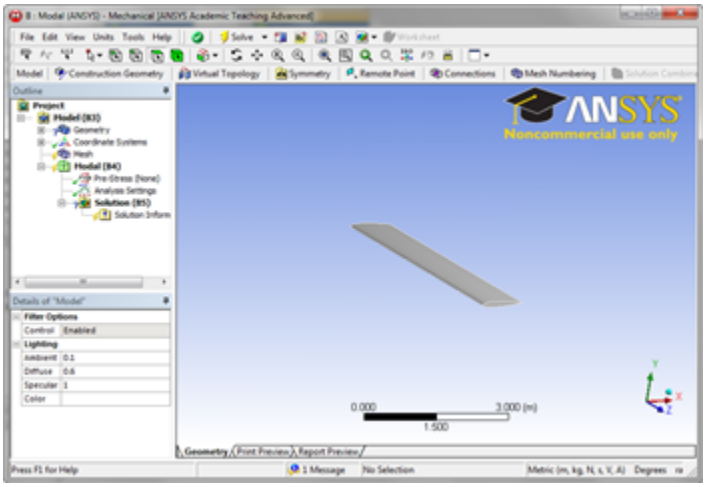
Modal Analysis of a Wing - Mesh

Author: Benjamin Mullen, Cornell University

Mesh

Open the Mesher

To open the mesher, double click the Model box  in the *Project Outline* window. This will load ANSYS Mechanical. You should now be able to see the airfoil geometry.



> **Meshing Warning**
>
> If you see an warning stating that the surfaces are higher order NURBS, ignore it: it simply says that creating the mesh may take a while to generate, but I've never had to wait more than a minute.

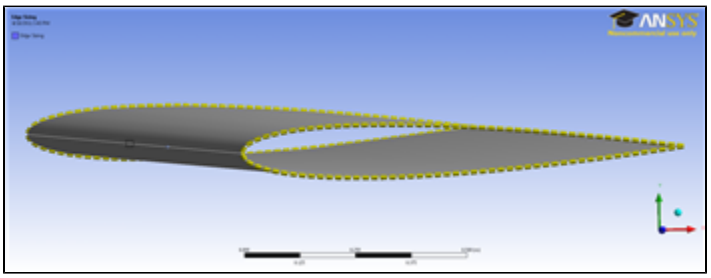
The first thing we are going to need to do when the mesher opens is specify the thickness of the airfoil walls. In the *Outline* window, expand ***Geometry*** and select ***Surface Body***. In the *Details* window, change the thickness to `0.01 m`. We also need to specify the material. In the *Outline* window. In the *Details* window, select ***Material > Assignment > Al 6061-T6***. The material has now been specified.

Mapped Face Meshing

To apply a mapped face meshing, first click on ***Mesh*** in the *Outline* window. This will bring up the Meshing Menu Bar at the top of the screen. Next, select **Mesh Control > Mapped Face Meshing**. Select the 2 faces of the mesh by holding down the left mouse button and dragging over the entire geometry. In the *Details* window, click ***Geometry > Apply*** - it should say 2 faces are selected.

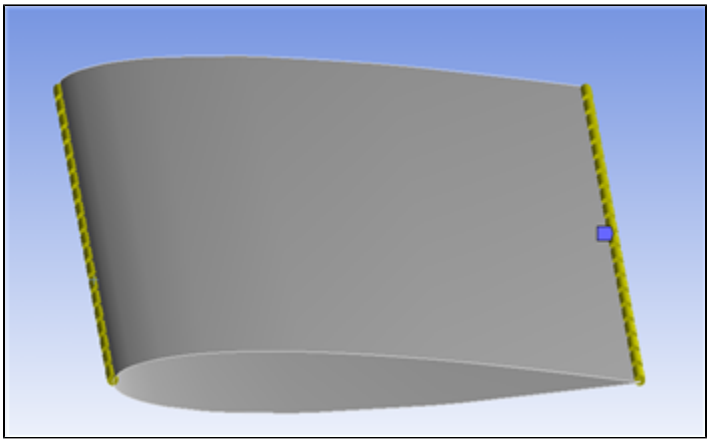
Edge Sizing

In the Meshing Menu, click **Meshing Control > Sizing**. Click the edge selection filter  . Select the 4 curved edges on the outside of the geometry that make up the shape of the NACA 0012 Airfoil as the picture shows:

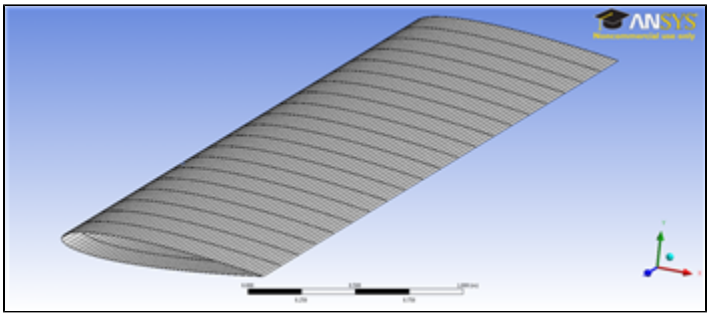


In the details window, select ***Geometry > Apply***, and select ***Type > Number of Divisions***. Change the ***Number of Divisions*** to `50`. Also, change ***Behavior > Hard***.

Next, create another Edge Sizing, and this time, select the 2 edges at the very front and very back of the airfoil that run along the wingspan, as the picture shows:



Again, in the *Details* window change the settings such that ***Type > Number of Divisions*** and ***Behavior > Hard***. This time, change the ***Number of Divisions*** to `20`. Generate the mesh by selecting **Mesh > Generate Mesh**.



Click here to enlarge

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules