

Wind Turbine Blade - Geometry

Author: Ben 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

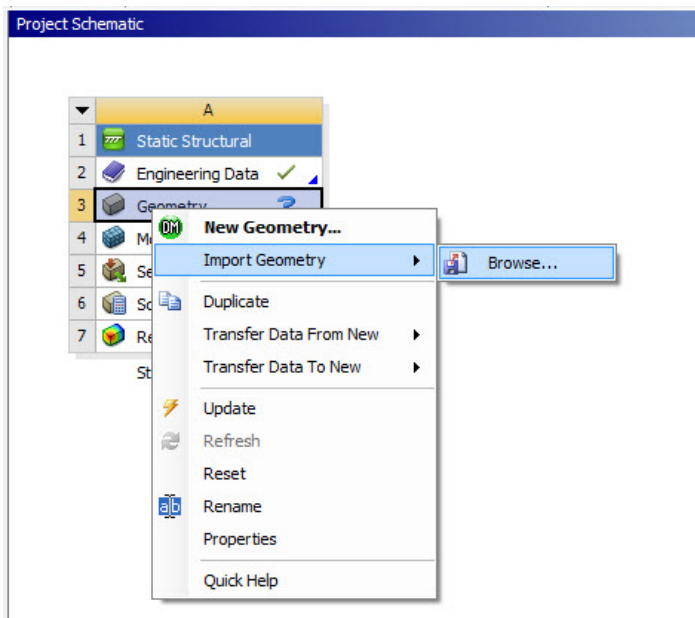
Geometry

 This tutorial is not being updated any more. We recommend that you follow [this newer tutorial](#) on fluid-structure analysis of a wind turbine blade. Thank you!

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

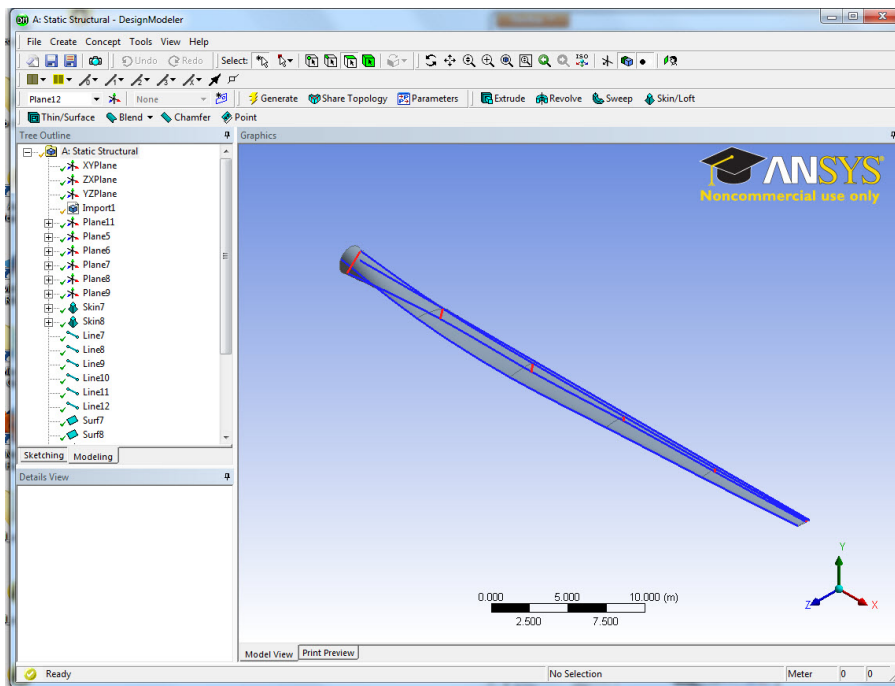
Downloading and Importing Geometry

Right click [here](#) and select **Save link as...** to download the geometry file. Save the file somewhere convenient, like your desktop or a working folder. In the *Project Schematic* page, right click on Geometry and import the downloaded file.



Visualizing the Geometry

Next, we will open the file to generate the geometry. Double click the imported geometry  **Geometry**  to open the Design Modeler. When the Design Modeler opens, a pop up window will ask us for the default units of measurement for the geometry. Select **Meter** and then press **OK**. The wind blade geometry will be displayed in the graphics window in Design Modeler. The blue lines highlight the shape of the wind blade and the spar and the red line highlights five sections of the blade/spar. No further work is needed.



Special Thanks

The original blade contained many geometric error when it was created in Solidworks. One major error was the connection between the wind blade and the spar. Thanks to Sean Harvey from ANSYS Inc., who has spent a great amount of work fixing the geometric errors on this wind blade.

Now that the geometry is imported and generated, we are ready to mesh the geometry.

[Go to Step 3: Mesh](#)

[Go to all ANSYS Learning Modules](#)