Wind Turbine Blade FSI (Part 1) - Geometry

Authors: Sebastien Lachance-Barrett (Cornell University) & Edwin Corona (University of Waterloo)

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.

We will save you some time and effort by providing you the geometry file for this tutorial. Please download the blade geometry right here. The blade was created right in Design Modeler and has the file extension .agdb which stands for DesignModeler Database.

Note: If you ever want to learn how to CAD your own wind turbine blade in SolidWorks ***(NOT the geometry for this tutorial)***, we recommend you take a look at this very nice guide! Credits to Nathaniel Gilbert, former Cornell MAE M.Eng student.

Overview of Current Geometry

In the following video we show you how to import this file and we explain important features of this geometry.

Summary of the above video:

1. Import Geometry
    a. Right click on Geometry > Import Geometry
    b. Browse > Select file in directory
2. Open Design Modeler
    a. Double click Geometry in project
    b. To get rid of ghost image, change view setting to wireframe, then back to shaded edges

---

Body Transformations

In this video, we show you how to correctly orient the blade given the pitch angle specification and we show you how to translate the blade so that the global axis becomes at center of rotation (center of hub).

It is not necessary to have strong background in wind power technology to follow this tutorial since its main function is to demonstrate how to perform an FSI simulation in ANSYS. However, having a basic understanding of wind turbine blade design and aerodynamics is suggested. It would be a good idea to know why blades are twisted for example. The following video starts with a brief statement on how blades are oriented but it might be difficult to understand if you have never been exposed to this material. I found two good brief introductions on wind turbine blade design and I strongly suggest you spend a few minutes learning the basics of this fascinating topic.

LearnEngineering - Wind Turbine Blade Design (Basic introduction; the video is very good)

Schubel, P., & Crossley, R. (2012). Wind Turbine Blade Design Review. *Wind Engineering,* 365-388. (A more thorough introduction if you are interested in learning a bit more about this topic)

Summary of steps in the above video:

1. Rotation of the blade
    a. Select body transformation -> rotate
    b. For Bodies, select the blade part (it select all surface bodies in that part)
    c. Axis Selection: x-axis, make it point in the positive direction, display plane first
    d. Angle is -70 degrees for it to be straight, be we want 4 degree offset at the tip so let's put -66 degrees
2. Translate so that global coord. is at the middle of the root
    a. Translate, choose our blade part for the bodies
    b. Direction definition: Coordinates
    c. Try to put it at the center, I found 0.9 in the y and 0.6 in the z to work well for this case.

3. Translation for hub
    a. Select the whole blade,
    b. Translation of 1m in the –x direction to account for hub

---

Defining the Blade Volume

In the following video, we close off all opening then make a solid body to represent the volume enclosed by the blade, out of the current surface bodies.

Summary of steps in the above video:

1. Closing the surfaces
    a. Create Surface from edges at the root to close this opening off
    b. Select this surface and all surfaces under part 5 and click form new part
2. Making a copy of our blade surface bodies to use later for the FEA.
    a. Translate
        i. Change preserve bodies to yes
        ii. Select all blade bodies (10)
        iii. Select any direction but specify the distance to be zero.
        iv. We now have two blades, rename this new blade, Blade FEA. Suppress the root surface from this blade (can hide the CFD Blade first). Hide Blade FEA.
3. Sew
    a. Suppress the spar bodies for Blade CFD (hide the part then show which ones are the spar)
    b. Body operation, Sew and choose only the outside surfaces.
    c. For create solid, click yes. Solid is created.

Drawing the Fluid Volume

We now draw the fluid volume.

Summary of steps in the above video:

1. Upwind sketch
    a. Select xy coord, new plane, offset z of 90m, generate
    b. New Sketch
    c. Make sure auto constraints cursor is selected
    d. Use arc by center tool
    e. Use line tool
    f. Dimensions angles to 60 degree each
    g. Dimension radius to 120 meters
2. Downwind sketch
    a. Same thing but place it 180 meters behind
    b. twice the radius, so 240 meters

Creating the Fluid Volume

Finally, we do a Boolean subtraction to remove the blade volume from the fluid geometry.

Summary of steps in the above video:

1. Skin
    a. Click skin, select both sketches
    b. Specify add material
2. Name this body, fluid, and specify that it’s a fluid
3. Boolean subtract
    a. Click Boolean, select target body to be the fluid body and tool body to be the blade
4. Show the blade inside with wireframe view

Go to Step 3: Mesh

Go to all FLUENT Learning Modules