Wind Turbine Blade - Mesh

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

Mesh

A This tutorial is not being updated any more. We recommend that you follow <u>this newer tutorial</u> on fluid-structure analysis of a wind turbine blade. Thank you!

Initial Setup

Close the Design Modeler if you haven't already, and open ANSYS Mechanical by double clicking Model When ANSYS Mechanical opens, notice that there is a question mark next to Geometry in the Project Outline - this means that there is something missing in this section. Expand *Geometry*, expand *Part 3*, and select any of the surface bodies.

Project		1
- 🚱 Mode		
	🕎 Part 3	
	🖳 ? 🔄 Surface Body	
	🖳 🤉 🔄 Surface Body	
	🖳 🤉 🔄 Surface Body	
	🖳 🤉 🛄 Surface Body	
	🖳 ? 🔄 Surface Body	
	2 Surface Body	
÷	Coordinate Systems	
Ē, 🕲 🤇	Connections	
	1esh	
ė, 🔁 🧐	Static Structural (A5)	
	Analysis Settings	
ė	🔞 Solution (A6)	
	Solution Information	

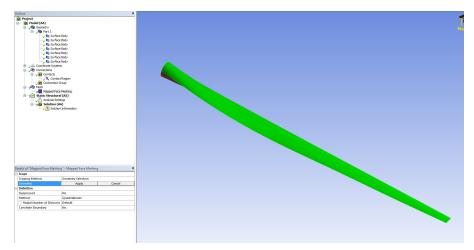
Notice that *Thickness* is highlighted as it does not have a value specified. Although we will ultimately specify a varying thickness for the the wind turbine blade, for now we will specify a dummy thickness so the geometry will mesh correctly. For this surface body, enter 1e-5 next to *Thickness*. Repeat with the same value for all other surface bodies.

비	Graphics Properties						
3	Definition						
	Suppressed	No					
	Stiffness Behavior	Flexible					
	Coordinate System	Default Coordinate System					
	Reference Temperature	By Environment					
	🗌 Thickness 🛛 🤇	1e-5					
	Thickness Mode	Refresh on Update					
ľ	Offset Type	Middle					
1	Material						
	Assignment	Structural Steel					
	Nonlinear Effects	Yes					
	Thermal Strain Effects	Yes					
Bounding Box							
Properties							
Statistics							

There should no longer be a question mark next to Geometry.

Mapped Face Meshing

Right click on *Mesh* and insert *Mapped Face Meshing* onto the top surface of the wind blade in isometric view. This will ask the meshing code to create hexahedral elements. ANSYS Mesher may create tetrahedral elements if Mapped Face Meshing is not used. In general, hex elements are more efficient that tetrahedral elements because it requires a smaller number of hex elements to mesh a model.



Repeat the step for the bottom surface. You may use the "Extend To Limit" tool to select all the faces instead of select them individually.



Body Sizing

Right click on Mesh and insert Sizing.

	22.
Project	
🗄 🗝 🞯 Model (A4)	
🖃 🗸 🕼 Geometry	
🖻 🗸 🆓 Part 3	
🦳 🗸 🔄 Surface Body	
······ Surface Body	
Surface Body	
Surface Body	
Coordinate Systems Connections	
Contacts	
Contact Region	
- Wald Mesh	
□¢∞ Mesh Insert	Method
	R Sizing
□ Sta	
Insert	R Sizing
□ Sta	Sizing K Contact Sizing A Refinement
Insert Insert Update Generate Mesh Preview	Sizing Contact Sizing
□ Sta Sta Generate Mesh □ Ka Show	Sizing Contact Sizing A Refinement
Insert Insert Update Generate Mesh Preview	 Sizing Contact Sizing Refinement Mapped Face Meshing Match Control
Insert Insert Update Update Generate Mesh Preview Show Create Pinch Controls	 Image: Sizing Image: Contact Sizing Image: Refinement Image: Mapped Face Meshing Image: Match Control Image: Pinch
Insert Update Update Generate Mesh Preview Show Create Pinch Controls Clear Generated Data	 Sizing Contact Sizing Refinement Mapped Face Meshing Match Control
Insert Insert Update Update Generate Mesh Preview Show Create Pinch Controls	 Image: Sizing Image: Contact Sizing Image: Refinement Image: Mapped Face Meshing Image: Match Control Image: Pinch
Insert Update Update Generate Mesh Preview Show Create Pinch Controls Clear Generated Data	 Image: Sizing Image: Contact Sizing Image: Refinement Image: Mapped Face Meshing Image: Match Control Image: Pinch

Right click in the graphics window and click on Select All.

Insert Go To Isometric View Set Set Set Q Restore Default Q Zoom To Fit	•	
 Zoom To Fit Cursor Mode View Select All 	;	

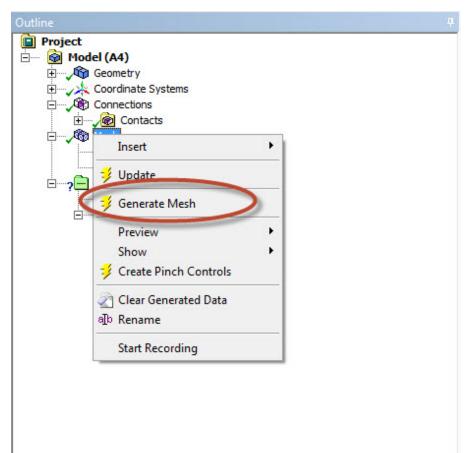
Change the *Element Size* to 0.2 m.

.

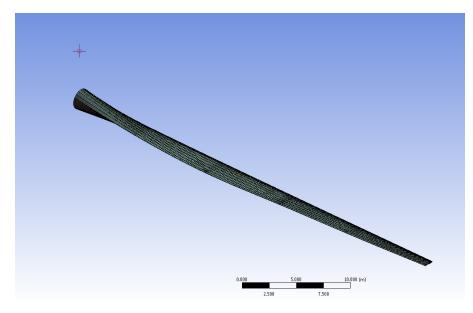
-	Scope		
	Scoping Method	Geometry Selection	
	Geometry	7 Bodies	
	Definition		
	Suppressed	No	
	Туре	Element Size	
	🗌 Element Size 🧹	0.2 m	
	Behavior	Sott	
	Curvature Normal Angle	Default	
	Growth Rate	Default	

Generate the Mesh

Right click on Mesh and click on Generate Mesh.



The meshed wind blade is shown below:



The meshed wind turbine blade consists of 7763 elements and 7566 nodes.

Now that the geometry has been meshed, we are ready to setup the physics controlling the simulation.

Go to Step 4: Physics Setup

Go to all ANSYS Learning Modules