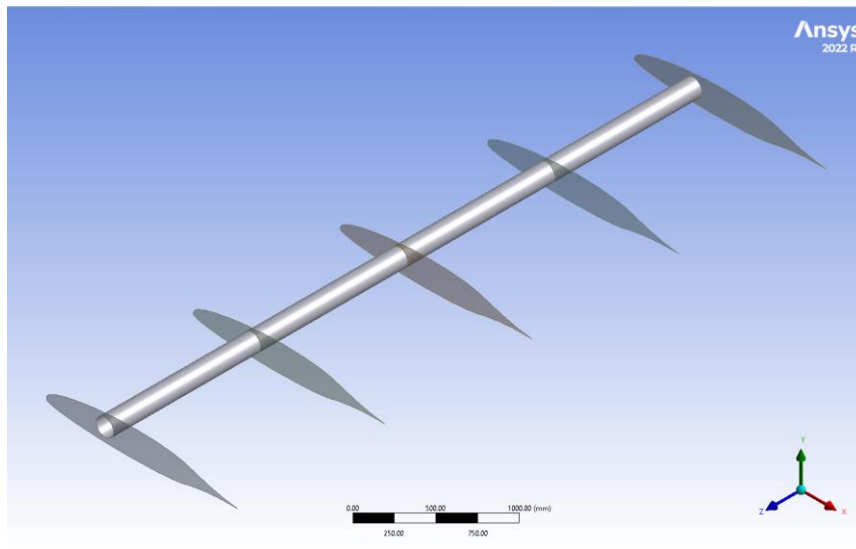# ENGSCI 344 Tutorial 5 – Part 2

In this tutorial, you will be assigning material properties, meshing, applying loads and boundary conditions to the wing geometry that you have built in tutorial 2.

The completed wing should look like the picture shown below. Note that you do not require a skin for this tutorial.
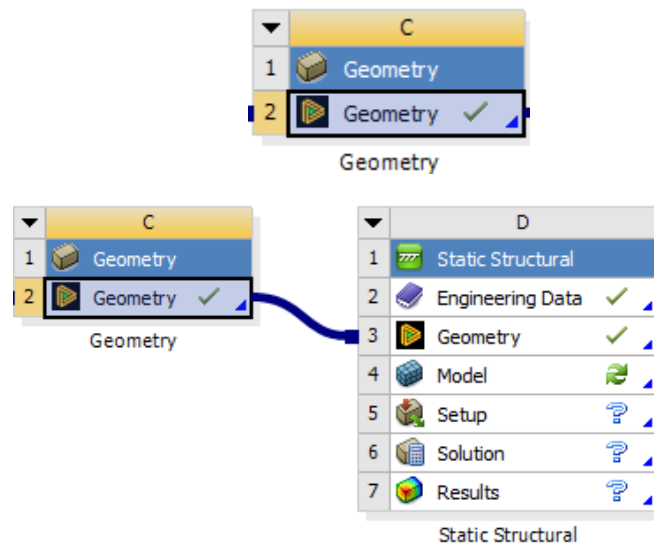

Creating a new material:

1. In the second tutorial, if you have created the geometry using the Static Structural Template, skip to step 3.
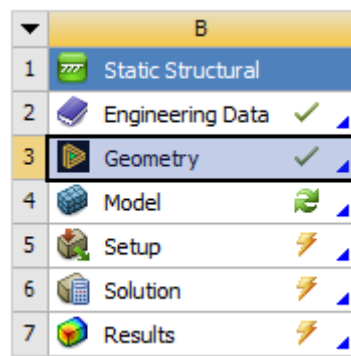
2. If you have created the geometry (wing geometry from tutorial 2) using the Geometry component system, link it to a new Static Structural Template.
   a. Create a new Static Structural Template.

b. To link one component system/template to another, click on the Geometry tab (left-click) of the Geometry Component System and drag it onto the Geometry tab of the Static Structural Template.

3. Using the steps outlined in part 1 of this tutorial, create a new material with the following material properties: Youngs modulus: 70 GPa, Poisson's ratio: 0.3. Note that you will need to use Al7075 for the project.

Creating a new part:

4. Open DesignModeler from the Geometry tab. The created wing should have multiple bodies (visible in the tree on the left), as shown in the figure below.
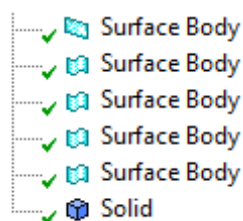
For the present problem, the interaction between different bodies (in this case the ribs and the spar) needs to be defined so that the different bodies and their mesh are properly joined together for the structural analysis to be accurate. One way of achieving this is by defining contacts and explicitly stating the type of interactions between all the bodies in ANSYS Mechanical. For some more complicated models this can be useful, but it adds significantly to analysis times, and can reduce the accuracy of stress calculations at interfaces. Instead, we will 'share the topology' between all the bodies such that the different bodies are considered as a single multibody part. This ensures that there is a continuous mesh across regions where different bodies touch (i.e. elements from different bodies share nodes). This is more efficient and accurate than using contacts.
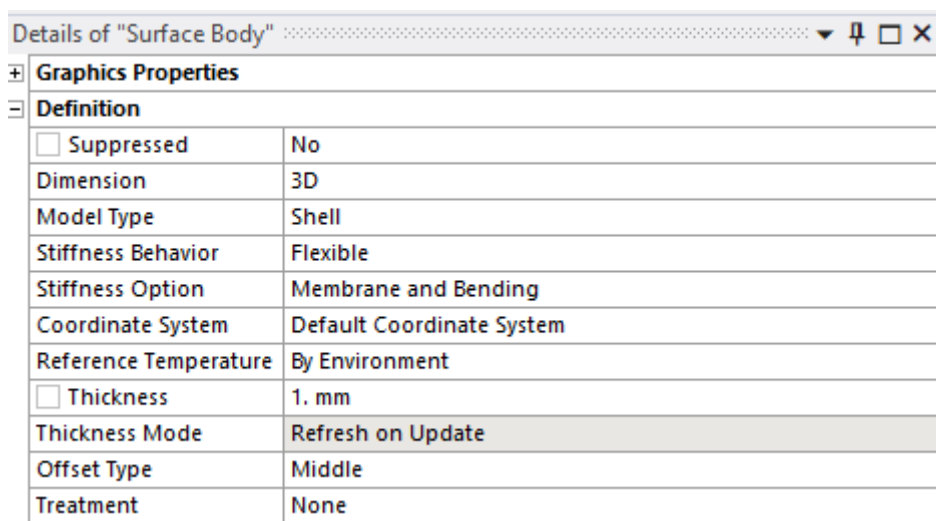
Have a look at *DesignModeler Help (press F1 in DesignModeler) > 3D Modeling > bodies and Parts > Parts > Shared Topology* for more details on shared topology.

5. Select all the relevant bodies of the wing, right-click and select Form New Part. This should group all the selected bodies into a new part.
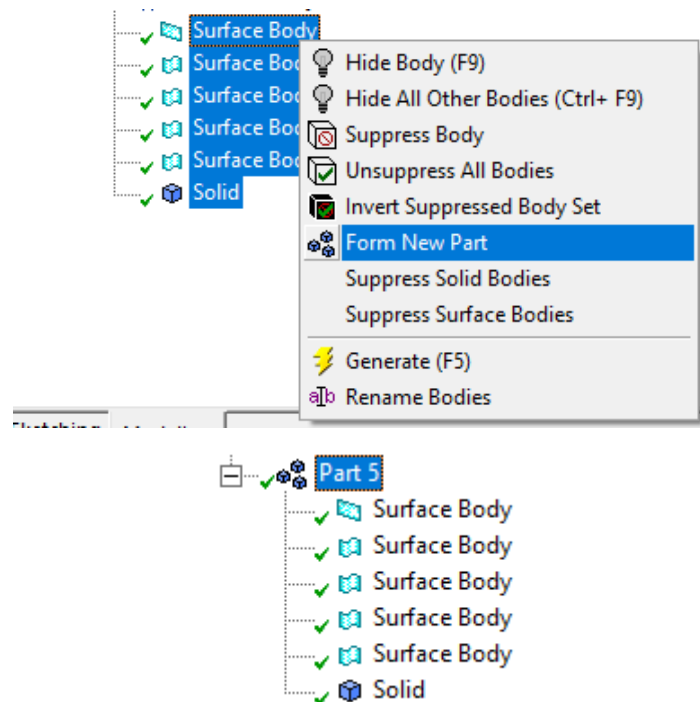
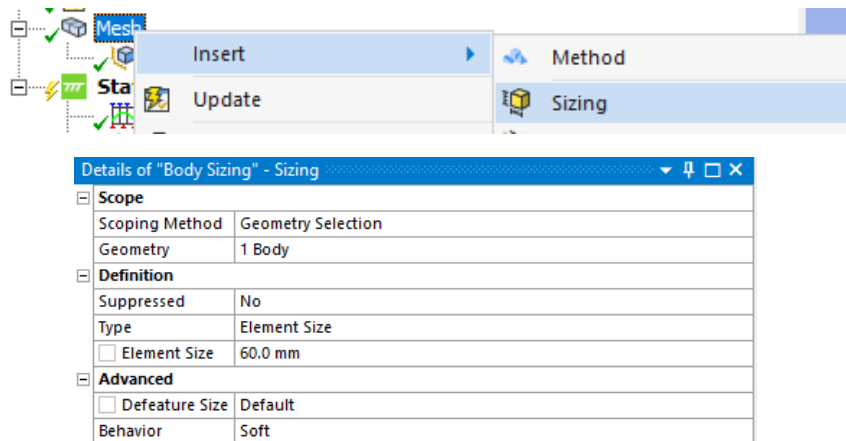# Applying loads, boundary conditions and solving the problem

6. Close DesignModeler and open the Model tab.
7. Assign the newly created material to all the ribs and the spar.
8. Set the thickness of the ribs to 1 mm (In the outline tree > Expand Geometry > Expand the newly created part > Select the ribs to bring the Details menu and set the thickness of the surface).

| Details of "Surface Body"	
⊞ **Graphics Properties**	
⊟ **Definition**	
☐ Suppressed	No
Dimension	3D
Model Type	Shell
Stiffness Behavior	Flexible
Stiffness Option	Membrane and Bending
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
☐ Thickness	1. mm
Thickness Mode	Refresh on Update
Offset Type	Middle
Treatment	None

9. In the meshing tab, change the global element size to 60mm, element order to quadratic and activate Adaptive Sizing.

| Details of "Mesh"	▼ � □ ×
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Quadratic
☐ Element Size	60.0 mm
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
☐ Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	5263.4 mm
Average Surface Area	4.8604e+005 mm^2
Minimum Edge Length	464.96 mm
Quality	

10. Create a new body sizing method for the entire spar using the parameters shown in the figures below (refer to Part 1 of this tutorial for help on local refinement).

Insert	▶		Method
Update			Sizing

Details of "Body Sizing" - Sizing	▼ � □ ×
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
☐ Element Size	60.0 mm
Advanced	
☐ Defeature Size	Default
Behavior	Soft

11. Mesh the model.
12. Apply a fixed support boundary condition to the base (left hand end) of the wing. Think about where it should be?
13. Apply a force load on the bottom edges of all the ribs. In the details tab of "Force", apply 1000N along the Y-axis and 150N along the X-axis. Is this a good approximation of the lift and drag forces? If not, how can it be improved for the project?
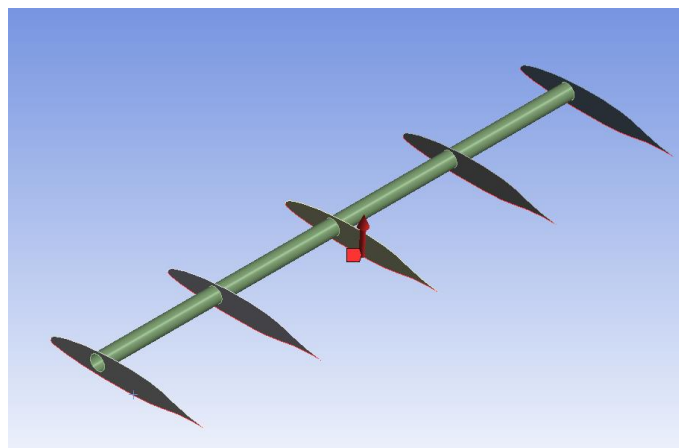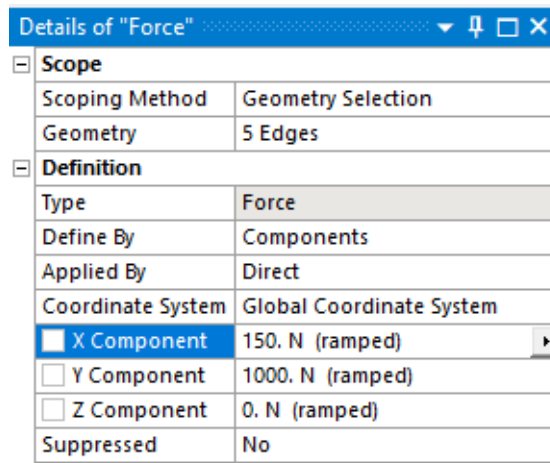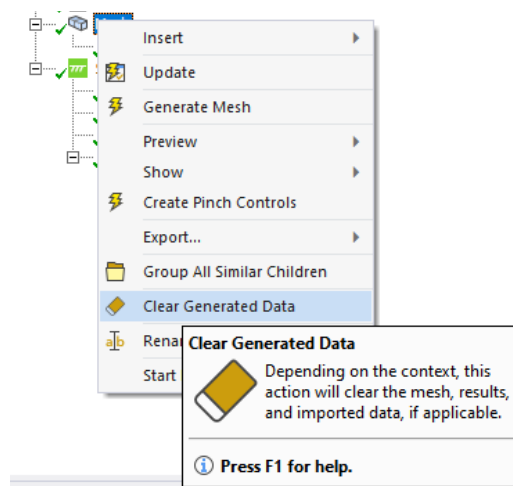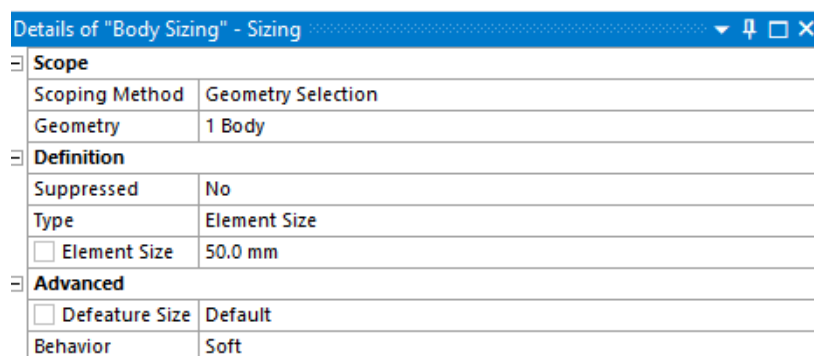
14. Solve the problem and record the maximum total deformation and the maximum equivalent stress.

Mesh refinement

15. Right-click on Mesh and clear the generated data.

16. Change the element size of Body Sizing to 50mm and solve the problem. Record the maximum total deformation and maximum equivalent stress.

17. Repeat steps 14 and 15 with a Body Sizing of 40mm and 30mm.

18. Plot the maximum total deformation and the maximum equivalent stress with respect to element size in MATLAB, Excel or Python. What do the trends in total deformation and equivalent stress tell you? Which quantity converges faster?

19. Are there any problems associated with the current mesh? If so, how can they be fixed?