ANSYS Composite Analysis for Characterization of the Project Aurora Fin Can

M. Nichitiu, C. Casebolt

MIT Rocket Team, Project Aurora

1 Fin

We begin with a description of the fin and its analysis.

1.1 Fin Structure, Characteristics

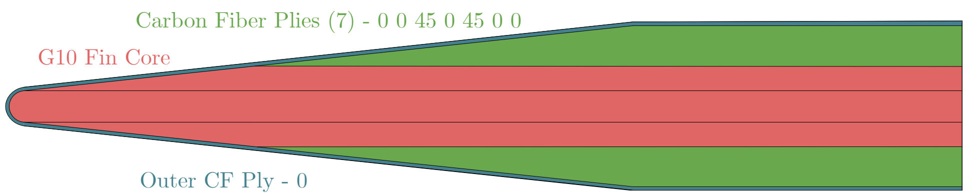

Figure 1. Side view of the composition of a fin.

Each fin is composed of a G10 core (thickness of 0.125”) followed by 7 plies of carbon fiber (CF) on each side[1]. Each CF ply has a thickness of 0.009”. The total thickness is thus

$$0.125\text{”} + 14(0.009\text{”}) = \mathbf{0.251\text{”}} \tag{1}$$

The CF plies have directionality; we chose to stack the 7 plies in the following orientations:

Ply	1	2	3	4	5	6	7
Orientation	0°	0°	45°	0°	45°	0°	0°

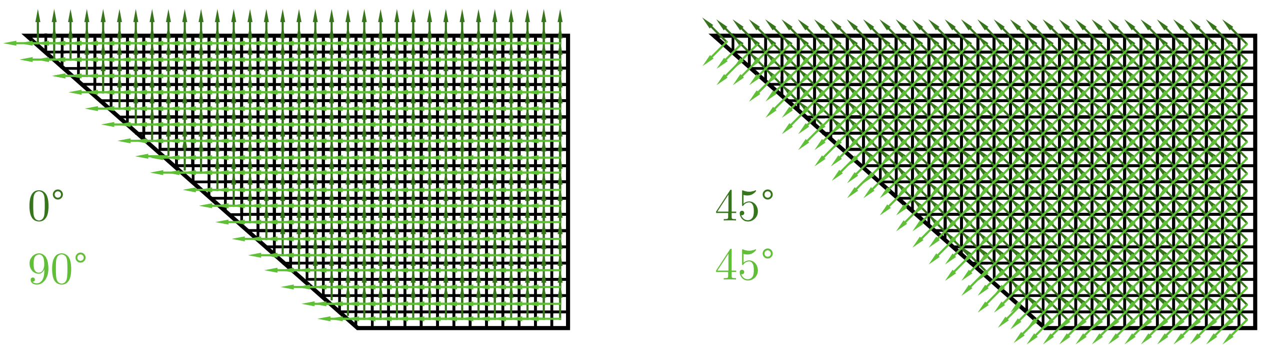

Figure 2. CF ply orientations: 0-90 and 45-45.

Once the CF 7-ply layup was complete, fillets were applied in order to smooth the leading, trailing, and outer edges; first, a 6° triangular cut was applied along the leading and outer edges, changing to 15° along the trailing edge. Then, fillet of radius 0.025” was applied along all edges. Technical drawings below illustrate this.

[1] There is a last layer of CF overwrap over the entire fin can exterior to act as protection and a smooth aerodynamic surface; this is discussed later.

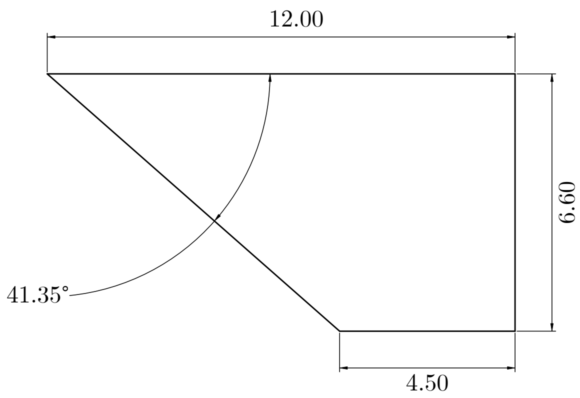

Figure 3. Dimensions of fin before fillets.

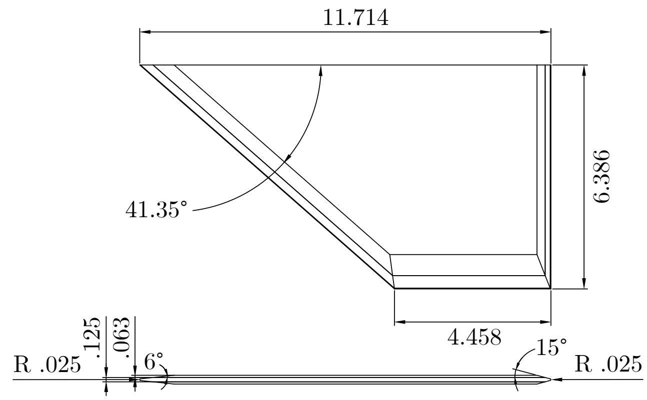

Figure 4. Dimensions of fin after fillets.

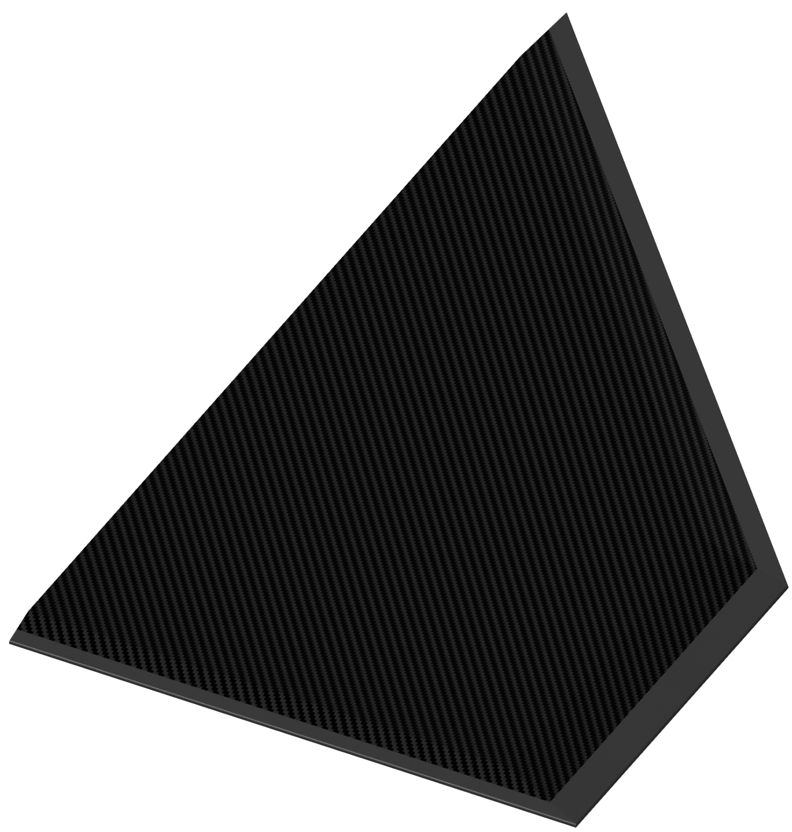

Figure 5. Render of a single fin, complete with layups and edge fillets.

1.2 Simulation Process

The process of simulation and determination of failure follows a few steps:

1. Use an **ACP Pre** block in ANSYS workbench.
2. Define the material properties.
3. Load Geometry using SpaceClaim.
4. Mesh in *Model*.
5. Define Composite Layups, export as *Solid*.
6. Transfer model(s) to **Static Structure**.
7. Define simulation parameters, then execute.
8. Obtain composite failure and material deformation.

Steps 1 and 2: Use an ACP Pre method in ANSYS workbench; Set the material properties.

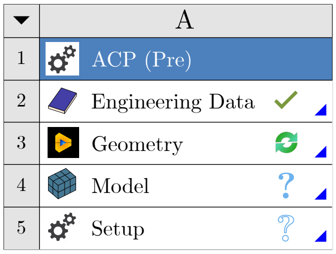

Figure 6. An **ACP Pre** Block.

We begin with the *Engineering Data* block. Three materials were defined: the Carbon Fiber fabric, G10 plastic, and the Epoxy resin. Their properties are notedbelow. It is essential to enter these data correctly in order to obtain realistic simulation results.

Carbon Fiber (CF) Definition

Property	Value	Unit
Density	1451	kg m^{-3}
Orth. Coeff. of Therm. Exp. ↓		
Coeff. of Therm. Exp. ↓		
X Coeff. of Ther. Exp.	$2.2 \cdot 10^{-6}$	°C^{-1}
Y Coeff. of Ther. Exp.	$2.2 \cdot 10^{-6}$	°C^{-1}
Z Coeff. of Ther. Exp.	$1.5 \cdot 10^{-5}$	°C^{-1}
Orth. Elasticity ↓		
X Young's Modulus	$5.916 \cdot 10^{10}$	Pa
Y Young's Modulus	$5.916 \cdot 10^{10}$	Pa
Z Young's Modulus	$7.5 \cdot 10^{9}$	Pa
Poisson Coeff: XY	0.04	
Poisson Coeff: YZ	0.3	
Poisson Coeff: XZ	0.3	
Shear Modulus: XY	$3.3 \cdot 10^{9}$	
Shear Modulus: YZ	$2.7 \cdot 10^{9}$	
Shear Modulus: XZ	$2.7 \cdot 10^{9}$	
Orth. Stress Limits ↓		
X Tensile	$5.13 \cdot 10^{8}$	Pa
Y Tensile	$5.13 \cdot 10^{8}$	Pa
Z Tensile	$5 \cdot 10^{7}$	Pa
X Compressive	$-4.37 \cdot 10^{8}$	Pa

Property	Value	Unit
Y Compressive	$-4.37 \cdot 10^{8}$	Pa
Z Compressive	$-1.5 \cdot 10^{8}$	Pa
XY Shear	$1.2 \cdot 10^{8}$	Pa
YZ Shear	$5.5 \cdot 10^{7}$	Pa
XZ Shear	$5.5 \cdot 10^{7}$	Pa
Orth. Deformation Limits ↓		
X Tensile	0.0092	
Y Tensile	0.0092	
Z Tensile	0.0078	
X Compressive	−0.0084	
Y Compressive	−0.0084	
Z Compressive	−0.011	
XY Shear	0.02	
YZ Shear	0.015	
XZ Shear	0.015	
Tsai-Wu Constants ↓		
XY Coupling	−1	
YZ Coupling	−1	
XZ Coupling	−1	
Ply Type ↓		
Type	Fabric	

G10 Definition

Property	Value	Unit
Density	2600	kg m^{-3}
Coeff. of Therm. Exp. ↓		
Coeff. of Therm. Exp.	$5 \cdot 10^{-6}$	°C^{-1}
Isotropic Elasticity ↓		
Derive from	Young & Poisson	

Property	Value	Unit
Young's Modulus	18.6	GPa
Poisson Coefficient	0.27	
Thermal Conductivity	1.27	W m^{-1} °C^{-1}
Specific Heat Pressure	802	J kg^{-1} °C^{-1}

Epoxy Resin (EPr) Definition

Property	Value	Unit
Density	1160	kg m^{-3}
Isotropic Elasticity ↓		
Derive from	Poisson & Shear	
Young's Modulus	$5.4824 \cdot 10^5$	psi

Property	Value	Unit
Shear Modulus	$2.0305 \cdot 10^5$	psi
Elastic Tensile Limit	7919.1	psi
Ply Type ↓		
Type	Isotropic	

Step 3: Load Geometry Using SpaceClaim.

Start by obtaining two STEP (for best compatibility) files from your chosen CAD software: a file containing only the G10 substrate, with the fillets applied, and then a file containing both the G10 substrate and CF layers, as separate *bodies* within the file. The first will be of immediate use; the second will be of use when adding the plies in order to match the realistic shape.

Begin by right-clicking cell **A3** in the ACP block, *Geometry*, and choosing *Import Geometry...* → *Search*, and choose your first STEP file (substrate only, one body). Once the action is successfully completed, right-click the cell once more and choose *Edit Geometry in SpaceClaim.* SpaceClaim will now launch.

Once in SpaceClaim, the sidebar on the left will contain a model tree, resembling the following:

↓ ACP-Pre
→ Body

In the main view, the STEP file will be displayed. In order to prepare the substrate for layup modeling, select the surfaces the CF will be bonded to (top and bottom faces), and copy-paste (`ctrl-C, ctrl-V`). The model tree will now resemble the following:

↓ ACP-Pre
→ Body
→ Surface
→ Surface

Name the surfaces according to their meaning: `top` and `bottom`, for example. Save your document (do not worry about path, ANSYS manages this for the user) by `ctrl-S`, and close SpaceClaim.

Step 4: Mesh using Mechanical

The next step is to right-click on cell **A4** in the ACP block, *Model*, and choosing *Edit*. In the tree on the left-hand side of the window that appears, search for the header of *Geometry*:

↓ **Model (A4)**
→ ✓ Geometry Import
↓ X Geometry
→ X ACP-Pre\Body
→ X ACP-Pre\Surface
→ X ACP-Pre\Surface
→ ✓ Materials
→ ✓ Coordinate System
→ ✓ Materials
→ ? Mesh
→ ✓ Named Selections

Choose the item *ACP-Pre\Body*, and in the lower window, choose the material associated with this body (G10) under the *Materials* header of the info box in the lower right. For the two surfaces, associate the material and also specify a thickness – as this is not physical, a small thickness $\left(10^{-10}\ \mathrm{m}\right)$ should be entered. Do not choose the object to be 2D, as this will prevent successful meshing and post processing.

Once the Xs are gone and replaced with ✓s, proceed to the *Mesh* item in the sidebar, and in the information box, choose under *Size* to **not** use an *adaptive sizing*; input an element size of **0.00195** in the *Element Size* box, just above the *Size* header.

When complete, click *Generate* in the toolbar to generate the mesh. Depending on your hardware, this may take 10s of seconds to a minute. When complete, save using `ctrl-S` and close Mechanical.