Trailing Edge Manufacturing for Project Zephyrus

Victoria Cure Hurtado and Shea Dahl

January 2026

1 Introduction

This document describes the manufacturing process for the aluminum trailing edges used on the Project Zephyrus fin assembly. The trailing edge design is based on the configuration presented in the Zephyrus AeroD CDR. The trailing edges are separate aluminum components that slot into the carbon fiber fins.

The purpose of the trailing edge components is to provide a clean, stiff, and manufacturable trailing edge profile for the Zephyrus fins. The design uses Aluminum 6061 and a 9.5 degree trailing edge profile. Each part includes a slot that interfaces with the carbon fiber fin. Two of the trailing edges span the full fin span, while the other two are shorter to leave clearance for the roll control surface.

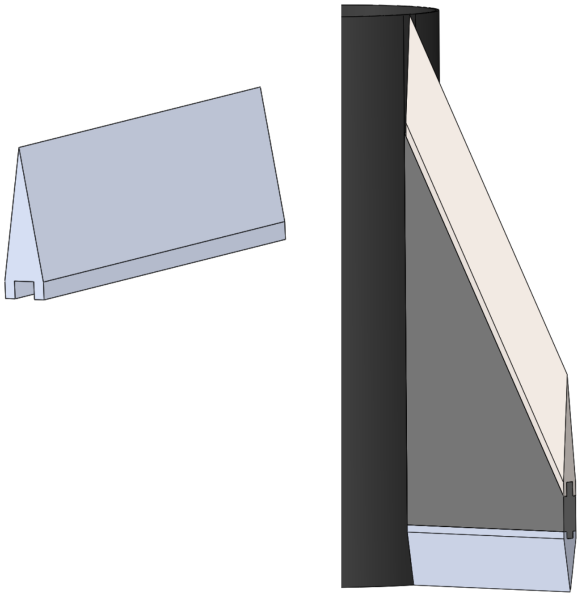

Figure 1: Trailing edge design presented in the Zephyrus AeroD CDR. The component uses a 9.5 degree profile, Aluminum 6061, and a slot that interfaces with the carbon fiber fin.

2 Part Overview

Four trailing edge components are manufactured for the fin assembly. The parts are similar in cross-section, but differ in length depending on whether they interfere with the control surface region.

Table 1: Trailing edge part variants.

Part Type	Description
Full-span trailing edge	Runs along the full span of the fin (6 in). Two full-span trailing edges are required.
Shortened trailing edge	Shortened to provide clearance for the control surface at the fin tip (5.25 in). Two shortened trailing edges are required.

Each trailing edge is machined from Aluminum 6061 bar stock. The final component has an angled trailing face, a flat interface surface, and a slot cut into the lower side of the part. The slot is used to locate the trailing edge onto the carbon fiber fin.

3 Materials and Tools

The required materials and tools are summarized in Table 2. Dimensions should be checked against the current CAD before machining.

Table 2: Materials, tools, and machines required for trailing edge manufacturing.

Item	Purpose
Aluminum 6061-T6 bar stock	Starting material for trailing edge components.
Mill	Primary machine used for facing, angled cutting, and slot cutting.
Face mill	Used to bring stock to thickness and clean large faces.
1/2 in end mill	Used for angled face machining.
1/4 in end mill	Used to cut the slot for the carbon fiber fin interface.
5 degree, 4 degree, and 1/2 degree angle blocks	Used to set the mill head angle to approximately 9.5 degrees.
Mill tram indicator	Used to set and return the mill head angle.
Dial test indicator	Used to verify part alignment before cutting.
Edge finder	Used to locate the part edges and center the slot cut.
1/2 in wrench	Used for mill head adjustment.
Parallels	Used to support the stock in the vise.

Estimated machining time is approximately 4 hours, depending on setup time, number of parts, and operator experience.

4 Manufacturing Notes

Before machining, confirm the latest CAD dimensions for both the full-span and shortened trailing edge parts. The two shortened trailing edges should be checked against the control surface geometry to ensure that the control surface has adequate clearance after installation.

The main manufacturing requirements are:

- Maintain the 9.5 degree trailing edge profile.
- Machine the part from Aluminum 6061-T6.
- Keep the slot centered and straight.
- Cut the slot to the specified depth.

- Deburr all edges after machining.
- Keep full-span and shortened parts clearly labeled after machining.

5 Stock Preparation

Begin with Aluminum 6061-T6 bar stock. The starting stock should be large enough to machine the full trailing edge profile and should be checked against the current CAD before machining.

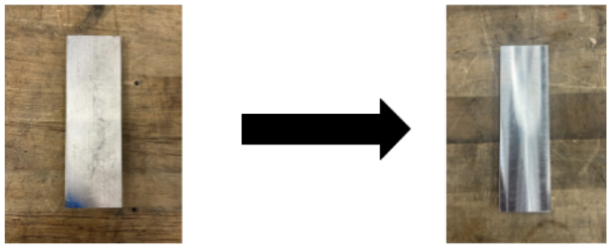

Figure 2: Before and after pictures of the base stock due to facing and sizing.

5.1 Facing the Stock

1. Insert the aluminum bar stock into the mill vise using parallels.
2. Orient the stock so that the largest face is directed upwards.
3. Use parallels that sit approximately 0.25 in below the top of the vise.
4. Use the face mill to make light passes on each side of the stock.
5. Bring the shortest dimension of the stock to approximately 0.500 in, with a tolerance of ± 0.005 in.
6. Recommended facing passes are approximately 0.020 in per pass.

5.2 Sizing the Remaining Sides

1. Rotate the stock so that the desired edge is facing upward.
2. Re-clamp the stock using parallels.
3. Use an end mill to make passes on each side as needed.
4. Bring the remaining sides of the stock to the desired blank dimensions.
5. Confirm the blank dimensions against the CAD or machining drawing before continuing.

6 Angled Face Cut Operation

The angled face cut creates the trailing edge profile. This operation requires tilting the mill head and carefully aligning the stock before cutting.

6.1 Zeroing the Mill Tram Indicator

1. Insert the mill tram indicator into the mill using a 1/2 in collet.
2. Press the tram indicator against the mill table with a slight preload, so that the dial indicators begin to rotate.
3. Zero both sides of the tram indicator by turning each dial until both hands point to the same value.

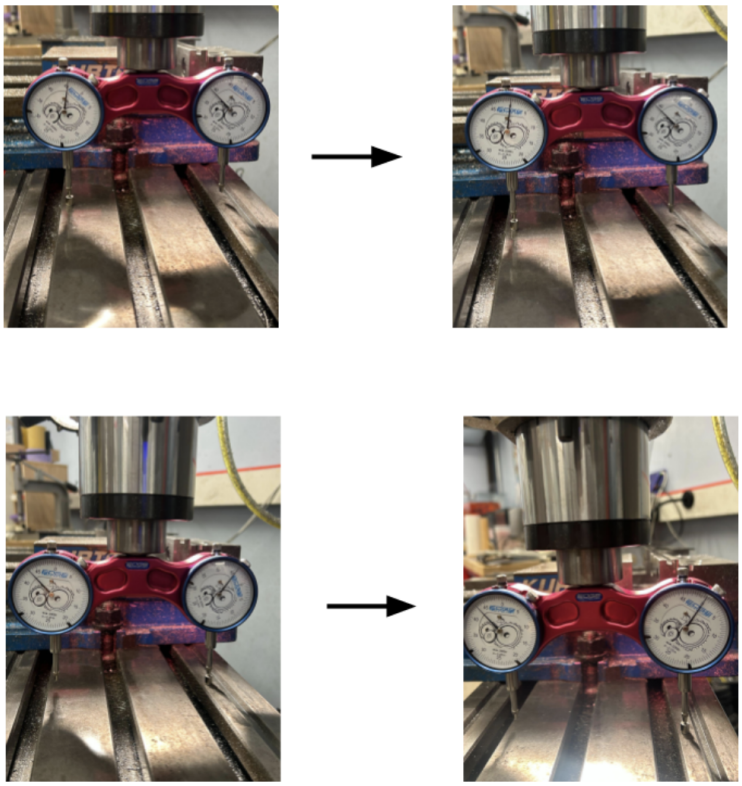

Figure 3: Mill tram indicator set to zero and checked using magnet.

6.2 Setting the Mill Head Angle

1. Stack the 5 degree, 4 degree, and 1/2 degree angle blocks and clamp them together.
2. Loosen the four front bolts on the mill head in a star pattern.
3. Rotate the mill head to the desired angle using the angle block stack as a reference.
4. Retighten the mill head bolts tightly in a star pattern.

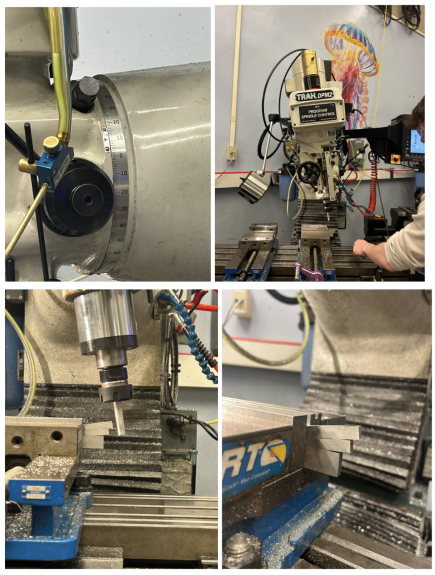

Figure 4: Mill tram indicator and angle block setup used to set the mill head angle for the angled trailing edge cut.

6.3 Touching Off and Tool Offset

Before cutting, touch off on the right side of the piece. Set the X-position such that the edge of the 1/2 in end mill lines up with the edge of the stock.

For a 1/2 in end mill with radius 0.25 in and a mill head angle of 9.5 degrees, the tool centerline offset is calculated as:

$$\text{Offset} = \frac{0.15}{\cos(9.5^\circ)} \tag{1}$$

This gives an offset of approximately:

$$\text{Offset} = 0.1521 \text{ in} \tag{2}$$

Offset the tool by approximately -0.1521 in before beginning the angled cut.

6.4 Clamping and Aligning the Stock

1. Clamp the stock into the vise and align it with the angle block.
2. Use parallels that sit approximately 0.25 in below the top of the vise.
3. Orient the stock so that the short sides are parallel to the vise edges. The stock should span the vise gap like a bridge.
4. Tighten the vise securely.
5. Clean the vise and parallels before every setup. Brush away all chips and particles before re-clamping.

6.5 Checking Alignment with the Dial Test Indicator

1. Attach the dial test indicator to the mill spindle.
2. Move the indicator to the surface that will be cut.
3. While holding the mill brake, preload the dial indicator by about one turn on the face to be cut.

4. Run the indicator along the surface to be cut.
5. The difference between the highest and lowest indicator readings should be within 0.005 in.
6. If the variation is greater than 0.005 in, loosen the vise, reorient the stock, retighten the vise, and repeat the alignment check.

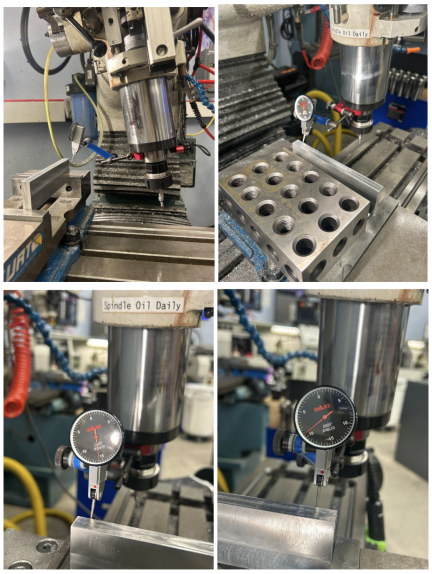

Figure 5: Part setup for the angled face cut. The part should be securely clamped and checked with a dial test indicator before machining.

6.6 Cutting the Angled Face

1. Confirm that the stock is tightly clamped.
2. Make angled face passes using the 1/2 in end mill.
3. Take no more than 0.020 in off per pass.
4. Continue making light passes until the desired angled profile is reached.
5. As the part approaches final geometry, physically measure the remaining stock instead of relying only on the mill readout.
6. When the top thickness approaches the 0.2000–0.2500 in range, measure the part after each pass.

6.7 Returning the Mill Head to Vertical

After the angled cut is complete, return the mill head to its original vertical position.

1. Reattach the tram indicator using the collet.
2. Loosen the mill head bolts in a star pattern.
3. Rotate the mill head back toward 0 degrees.
4. Press the ends of the tram indicator against the mill table.
5. Adjust the mill head angle until both dials give the same measurement.
6. Retighten the mill head bolts in a star pattern.
7. Take a picture of the mill re-zeroed to ensure completeness.

7 Slot Cut Operation

The slot cut allows the trailing edge to interface with the carbon fiber fin. The slot should be centered and cut to the specified depth.

7.1 Slot Setup

1. Clamp the trailing edge part into the mill vise.
2. To keep the part level, slide a 1/16 in jig piece into the setup as needed.
3. Use the edge finder to locate the middle of the part and zero the mill.
4. If the part thickness is approximately 0.495 in, the centerline location is found using half the part thickness plus the edge finder offset.
5. For this setup, the centerline offset is approximately 0.3475 in.

7.2 Cutting the Slot

1. Install a 1/4 in end mill.
2. Make slotting passes of approximately 0.0250 in per pass.
3. Continue cutting until the slot reaches a final depth of approximately 0.300 in.
4. Check that the slot is straight, centered, and free of excessive burrs.

Figure 6: Slot cut setup. The slot is machined with a 1/4 in end mill and is used to interface the aluminum trailing edge with the carbon fiber fin.

8 Troubleshooting

8.1 Lip Formation During the Angled Cut

If a lip begins forming during the angled face operation, the most likely causes are:

- A zeroing issue during touch-off at the start of the angled operation.
- The stock was not clamped securely enough in the vise.

- The tool was not positioned correctly relative to the stock edge.

If this happens, lower the mill slightly and repeat the same pass. For example, if the Z-position was set to 0.040 in, lower it to approximately 0.030 in or 0.020 in and check whether the lip begins to clean up. Make sure to stay within tolerance.

8.2 General Fixes

- As the angled cut approaches the final shape, physically measure the part often instead of relying only on the mill readout.
- Measure the thickness of the top of the trailing edge after each pass once the remaining thickness is in the 0.2000–0.2500 in range.
- If the issue appears to come from zeroing, re-check the touch-off location and tool offset before continuing.
- If the issue appears to come from clamping, stop machining, clean the vise, re-clamp the part, and re-check alignment with the dial test indicator.

9 Finishing and Inspection

After machining, deburr all edges of the trailing edge part. Pay particular attention to the slot edges, the angled trailing face, and the ends of the part. Remove any sharp edges that could damage the carbon fiber fin during installation or handling.

Inspect each part for the following:

- Correct part length for full-span or shortened trailing edge variant.
- Clean 10 degree trailing edge profile.
- Slot centered on the part.
- Slot depth approximately 0.300 in.
- No major chatter, gouges, lips, or surface defects.
- Part fits onto the carbon fiber fin without excessive force.

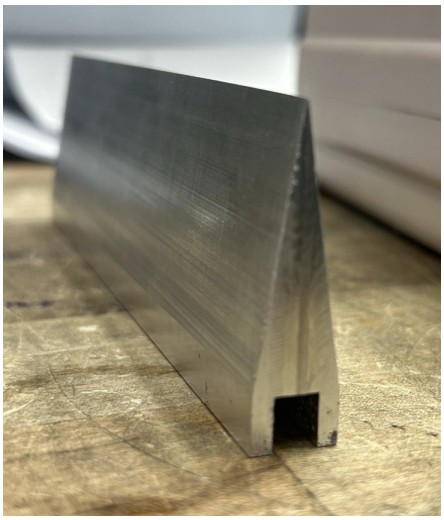

Figure 7: Finished aluminum trailing edge component after machining and slot cutting.

References

1. MIT Rocket Team, AeroD Subteam. *Zephyrus AeroD CDR*. Critical Design Review slide deck, December 2025.

2. MIT Rocket Team, AeroD Subteam. *Trailing Edge Manufacturing Procedure*. Internal manufacturing notes, January 2026.