

To: Professor Willy
From: Emilio Huggins- Active Rocket Controls
Date: 2/2/2026
Subject: Self Learning Memo- ANSYS CFD

1) Purpose and Skill Chosen

For this self learning assignment, I have decided to use ANSYS to simulate airflow over a fin of our rocket to find drag force. This is a valid self-learning topic because I did not have prior proficiency in setting up CFD cases in ANSYS for external flow and had to learn the practical workflow: identifying the correct boundaries, applying the proper boundary conditions, initializing the solver, diagnosing nonphysical results, and verifying that the reported forces and contours are meaningful.

2) Skills Developed

Working through ANSYS CFD improved several design-related skills that apply directly to our capstone project. First, it strengthened my modeling and simulation ability, turning a geometry and test conditions into an actual computational case. Second, it forced an iteration and troubleshooting mindset, since small setup mistakes (like selecting the wrong surfaces or leaving a boundary value at zero) can completely change the results. Third, it built a verification mindset: I learned to check the flow field visually and confirm that reports/monitors are using the right surfaces before trusting any numbers. Finally, it improved technical communication, since CFD results need to be shown through screenshots and plots that clearly support what was done and what was learned.

3) Self-Learning Plan

3.1) Learning Objectives

Objectives were established in order to track skill development. Some such objectives were correctly defining computational domain boundaries, apply inlet/outlet conditions and initialize a solution that produces an actual flow, generate velocity magnitude and pressure contours showing proof, and extracting force reports of aerodynamic forces on fin surfaces.

3.2) Evidence of Self-Learning

Progress is documented through screenshots and saved solver setups showing:

- Boundary views with outer walls toggled off to isolate fin surfaces.
- Boundary condition panels showing the inlet velocity magnitude applied.
- Initialization and iteration output showing nonzero solver activity.
- Flow contours showing velocity magnitude across the domain.
- Force report windows showing initial zero-force outputs (used as a diagnostic checkpoint).

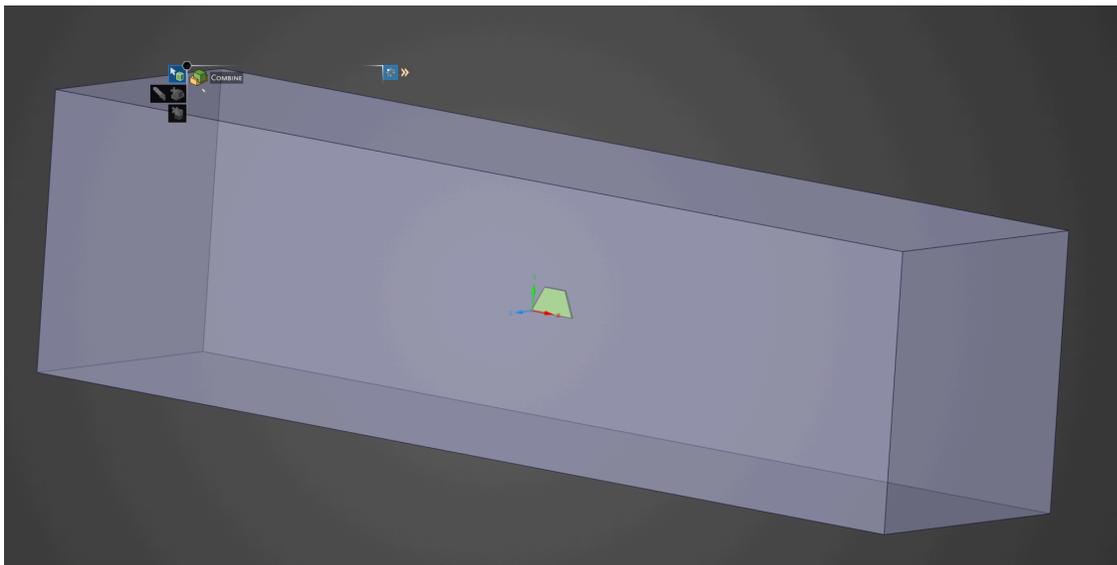


Figure 1: Fin in Air Enclosure Before Cut-merge

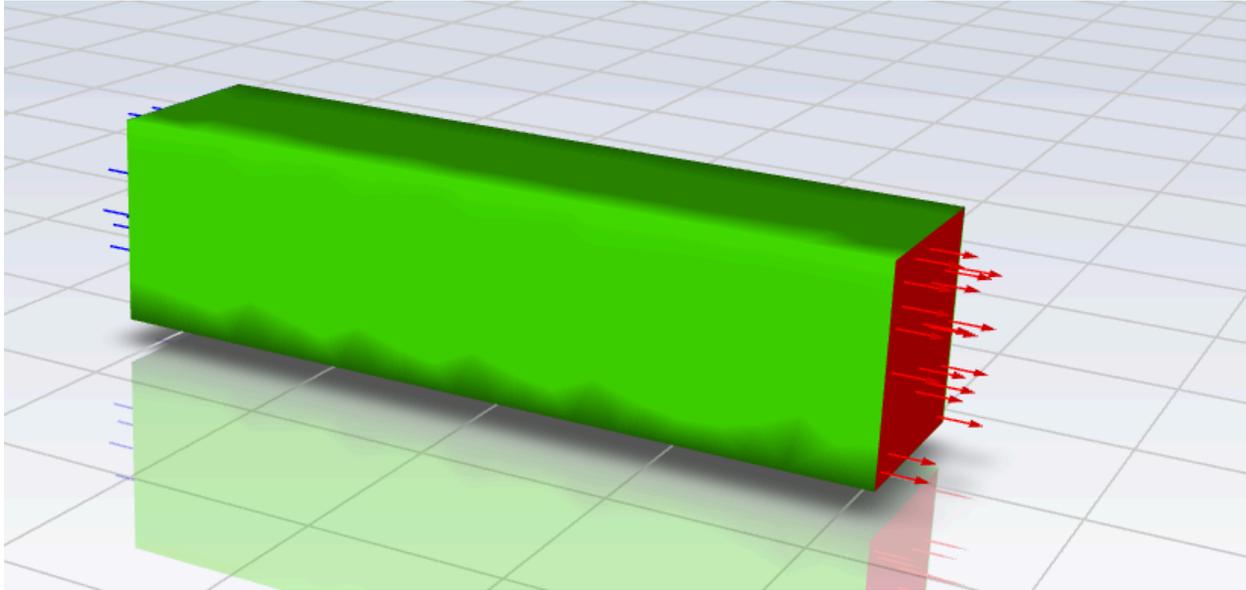


Figure 2: Finished “Air Tunnel” With Inlet and Outlet BC

Calculation complete.

Forces	Forces [N]		Coefficients		Total	
Zone	Pressure	Viscous	Pressure	Viscous	Pressure	Viscous
Viscous						
wall-enclosure_enclosure	(0 6.8212103e-13 -1.4551915e-11)	(0 1.113667e-12 -2.3758229e-11)	0	0	0	0
(0 0 0)						
Net	(0 6.8212103e-13 -1.4551915e-11)	(0 1.113667e-12 -2.3758229e-11)	0	0	0	0
(0 0 0)						
Forces - Direction Vector	(1 0 0)					
Zone	Forces [N]	Viscous	Total	Coefficients	Viscous	Total
wall-enclosure_enclosure	Pressure	0	0	Pressure	0	0
Net	0	0	0	0	0	0

Figure 3: Non-zero Solver Activity

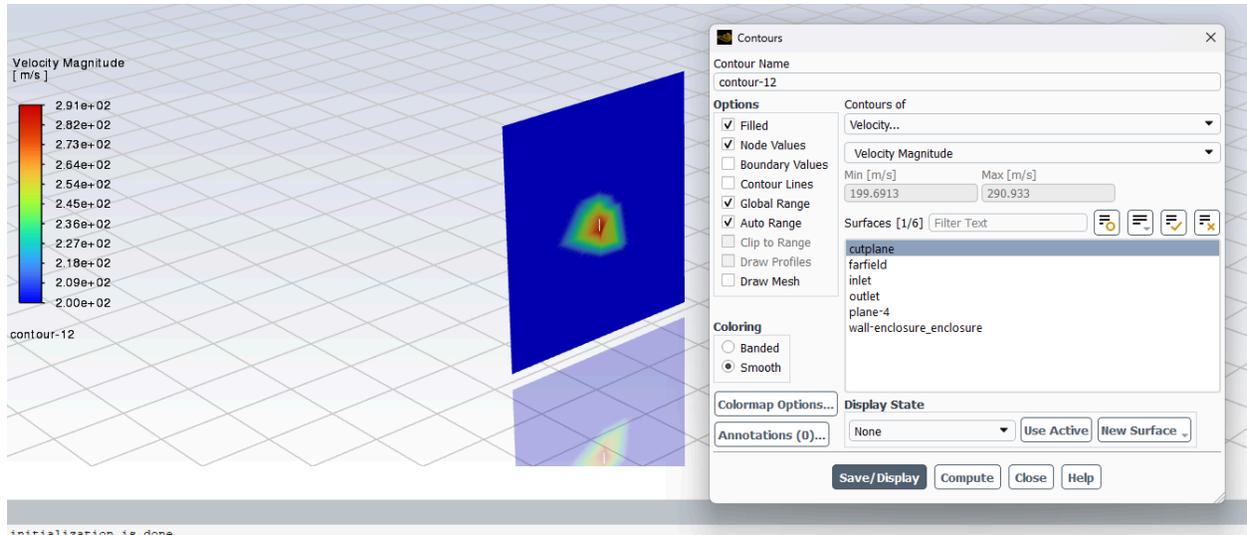


Figure 4: Velocity Magnitude Contour

Forces						
Zone	Forces [N]			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
Viscous						
wall-enclosure_enclosure	(0 0 0)					
	(0 0 0)		(0 0 0)			(0 0 0)

Net	(0 0 0)		(0 0 0)	(0 0 0)		(0 0 0)
	(0 0 0)		(0 0 0)			
Forces - Direction Vector (1 0 0)						
Zone	Forces [N]			Coefficients		
	Pressure	Viscous	Total	Pressure	Viscous	Total
wall-enclosure_enclosure	0	0	0	0	0	0
Net	0	0	0	0	0	0

Allocating 3x1=3 pixel map.						
0 pixels filled, area = 0						
Area of surface (3) projected onto plane (1, 0, 0): 0						

Figure 5: Initial Zero-Force Output

4) Work Completed So Far (Chronological Workflow)

4.1) Geometry and Boundary Identification (Major Initial Challenge)

The first major obstacle was correctly identifying and selecting the intended fin wall surfaces within the enclosure while avoiding accidental selection of farfield/exterior faces. During setup, clicking on what visually appeared to be the “inside” fin walls consistently highlighted outside surfaces instead, making it difficult to confidently assign boundary types to the correct faces.

To address this, I used visibility controls (hiding faces and toggling display options) to isolate boundary groups. By selectively displaying only certain boundaries (for example, temporarily disabling outer walls display), I was able to reduce clutter and better confirm which surfaces belonged to the fins versus the surrounding domain. This step was necessary before any force reporting could be meaningful.

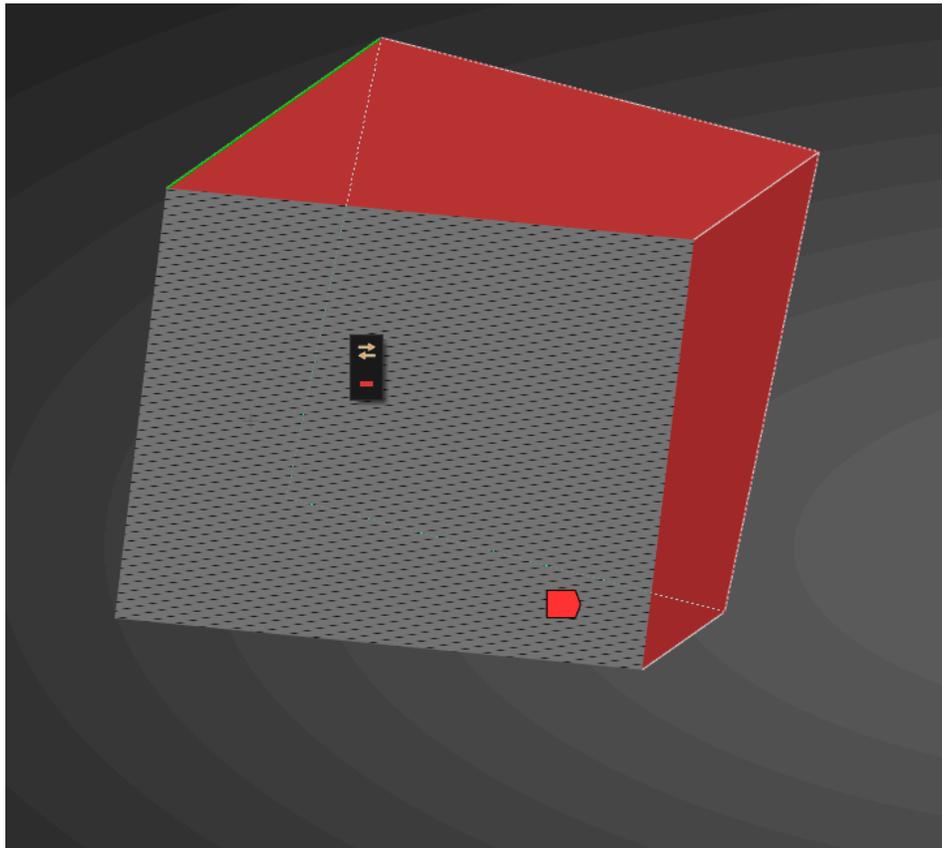


Figure 6: Sliced Geometry With Boundary Surface Selection

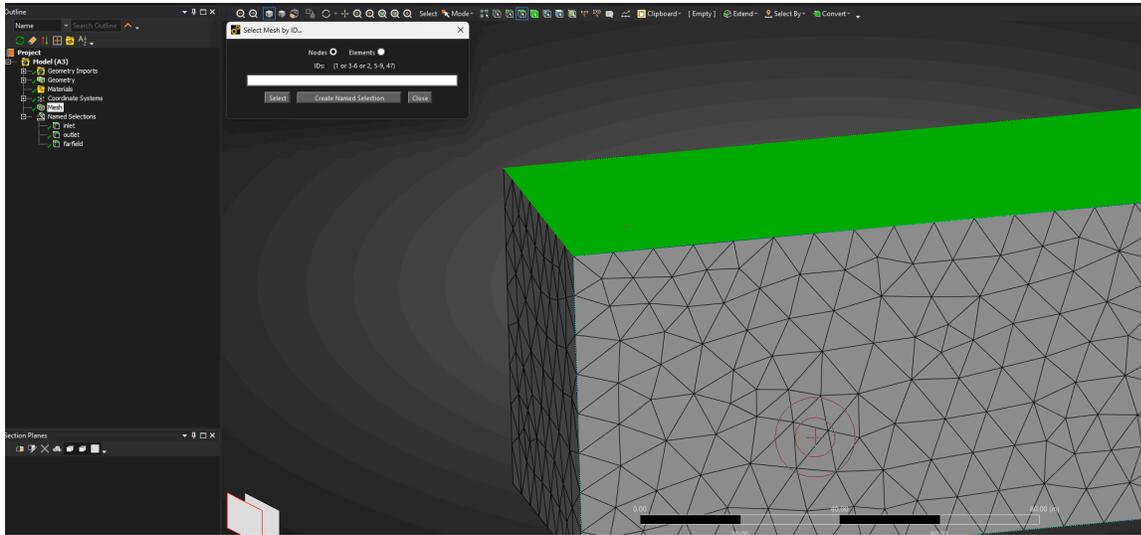


Figure 7: Meshed Enclosure with Boundary Condition Tree

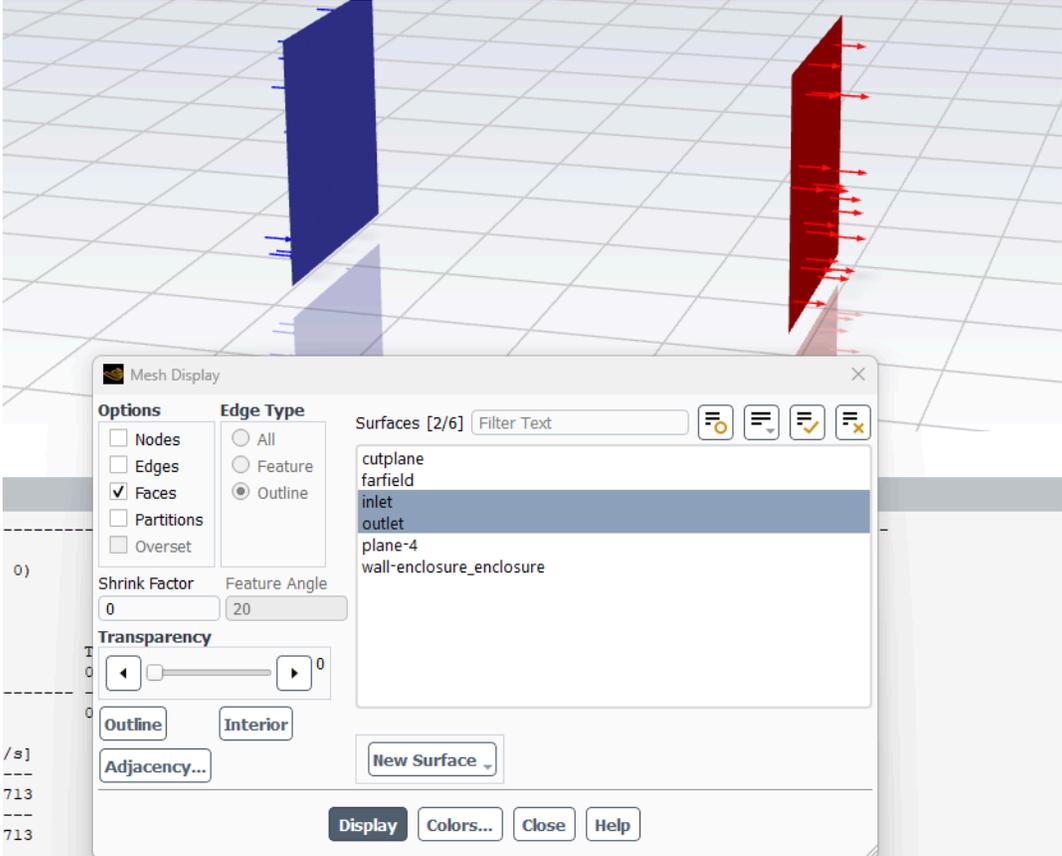


Figure 8: Inlet / Outlet Boundary Condition Surfaces

Learning outcome: Surface selection and boundary grouping are “make-or-break” steps in CFD. If the fin walls are not separated correctly, downstream force and pressure results can be wrong.

4.2 Boundary Conditions and Initialization

After defining surfaces, the next issue was that output values (especially forces) were initially reading as zero. Diagnosing suggested the solution was not being worked properly because the inlet condition effectively resulted in no flow. I found that the inlet velocity was set to 0, which naturally produced zero forces and a trivial flow field.

To correct this, I applied a nonzero inlet velocity (using 200 m/s as a test condition) and reinitialized the solver. After doing this, solver iteration output began producing nonzero values, indicating that the model was now generating an active flow field rather than remaining stagnant.

Learning outcome: Boundary conditions must be verified numerically, not assumed. A case can seem “configured” while still producing a no-flow solution.

4.3 Flow Visualization Check (Confirming Nonzero Flow Field)

To confirm the applied inlet condition was producing meaningful flow behavior, I generated a velocity magnitude contour. This served as a quick verification step to ensure the domain contained a nonzero flow field and that the simulation was producing gradients around the fin geometry rather than remaining at uniform zero velocity.

This contour confirmed that the solver was producing flow behavior, and it provided a baseline output even before force reports were trustworthy.

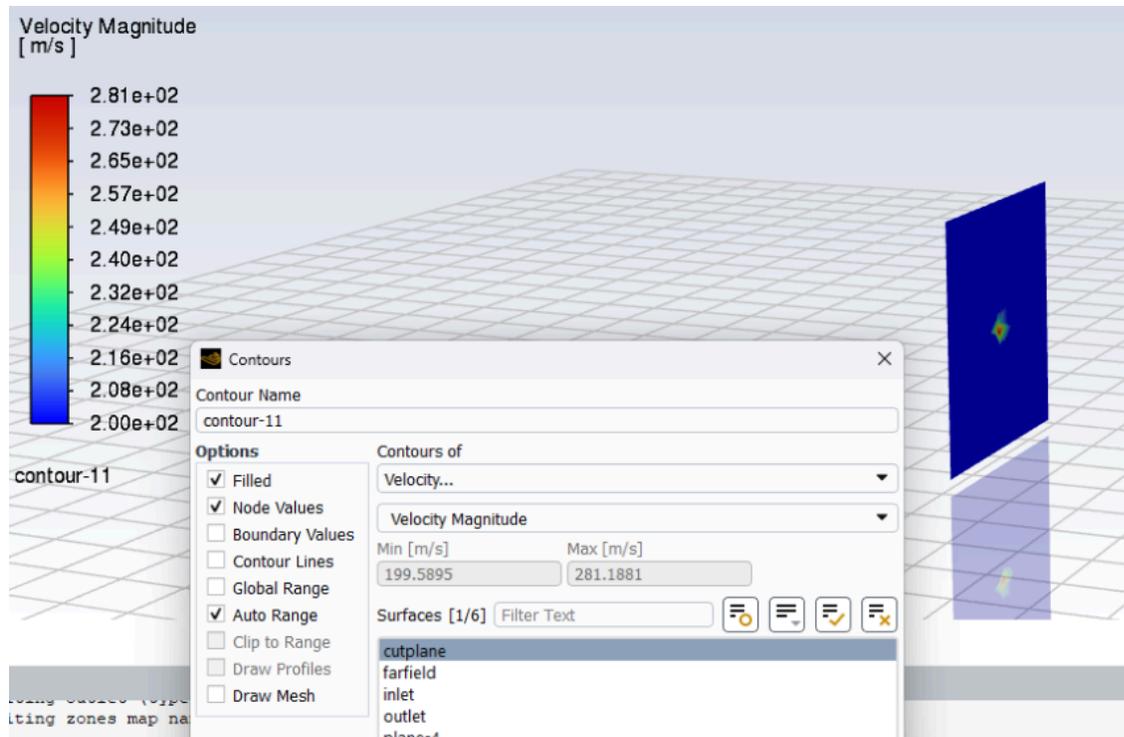


Figure 9: Velocity magnitude contour plot showing nonzero flow field

Learning outcome: Flow visualization is a required “sanity check.” It verifies that the solver is not stuck in a trivial solution and helps detect obvious setup errors early.

4.4 Current Force Results

After correcting the inlet velocity from 0 to 200 m/s and reinitializing, the solution began showing nonzero iterations and produced a nonzero flow field (verified with a velocity magnitude contour). Initially, however, the force reports still returned zero, hinting the reporting setup or surface selection was not yet correct.

Later in the troubleshooting sequence, the solver began reporting a non-zero drag force. While this is progress, the drag value appears suspicious and is not considered validated. Given earlier boundary-selection problems (inner fin walls were difficult to select cleanly without accidentally selecting farfield surfaces), the most likely explanation is that the drag report is being influenced by an incorrect surface set, incorrect boundary typing, or an incomplete verification of reference directions.

For this self-learning assignment, the current “final” result is that the simulation produces a non-zero drag force, but it is not yet validated and is being treated as a preliminary/likely incorrect output.

```

delete.

Forces [N]
Pressure
enclosure (0 6.8212103e-13 -1.4551915e-11)
           (0 1.113667e-12 -2.3758229e-11)
-----
Total
           (0 6.8212103e-13 -1.4551915e-11)
           (0 1.113667e-12 -2.3758229e-11)
-----
Coefficients
Pressure
enclosure (0 1.113667e-12 -2.3758229e-11)
           (0 1.113667e-12 -2.3758229e-11)
-----
(0 6.8212103e-13 -1.4551915e-11)
           (0 1.113667e-12 -2.3758229e-11)
           (0 6.8212103e-13 -1.4551915e-11)
           (0 1.113667e-12 -2.3758229e-11)

on Vector (1 0 0)
Forces [N]
Pressure
enclosure 0
           0
           0
-----
           0
           0
           0
-----
Coefficients
Pressure
enclosure 0
           0
           0
-----
           0
           0
           0
-----
Viscous
Total
Coefficients
Pressure
Viscous
Total

```

Figure 10: Initialization / iteration output showing nonzero iteration activity

Learning outcome: CFD can produce numerical outputs that look “real” even when setup details are still wrong. Having a non-zero result does not conclude correctness, additional verification is required.

5) Application to Capstone

This self learning effort directly supports our capstone project because CFD is intended to provide aerodynamic loads needed for design decisions, including fin structural sizing and justification of safety margins. Even at the current stage, building a functional CFD pipeline (geometry → boundaries → BCs → solve → report) is valuable because it establishes the process we will use to generate credible aerodynamic loads later.

For the ARC rocket, the intended use of CFD outputs includes:

- Estimating aerodynamic forces on fins to support bending/attachment load calculations
- Evaluating pressure/velocity behavior around fin geometry at scenario flight conditions
- Supporting design choices with simulation visuals and quantified results in design reviews

At the moment, the drag result is treated as preliminary and suspicious, meaning it is not yet used as a final design input. The primary value right now is confirming the workflow is close to producing extractable forces, and identifying what still needs verification.

6) Results

At this stage, the simulation workflow has progressed from a stagnant case (inlet velocity unintentionally set to 0, force reports all zero) to an active case (inlet velocity set to 200 m/s, solver producing nonzero field behavior, and eventually reporting a non-zero drag force). However, due to earlier selection ambiguity between fin walls and farfield surfaces and because the drag result appears inconsistent with expectations, the reported drag is not yet trusted as a design input.

For the purpose of this self-learning memo, the current status is:

- A working CFD pipeline is close to being established (geometry → BCs → solve → report).
- A non zero drag force is produced, but it is suspected to be wrong.
- The immediate plan is to document the workflow and return later to diagnose.

7) Next Steps

With further opportunity for more in-depth diagnosis, the next steps are:

1. Confirm the force report surface list contains only the intended fin walls (not farfield/enclosure faces).
2. Verify the fin surfaces are typed as wall boundaries.
3. Check force direction axes and reference settings.
4. Add force monitors vs iteration to confirm forces update and stabilize.
5. Run a simplified baseline case to confirm the pipeline produces physically reasonable drag trends.