CFD Analysis and Experimental Design of Serpentine Air-Cooling for Formula Student Battery Pack

BY RAHUL OM (1RV23AS405)

1. Abstract

This report details the design, simulation, and analysis of a serpentine air-cooling system for a Lithium-ion battery pack intended for a Formula Student Electric Vehicle (FSEV). Based on established literature, a 18-cell module using Melasta pouch cells was modeled in CAD and analyzed using Computational Fluid Dynamics (CFD) in ANSYS Fluent and Fusion 360. The study focused on evaluating flow uniformity, pressure drop, and thermal distribution at a high inlet velocity of 17m/ s. Results indicate a serpentine flow path successfully routes air through the module with a significant static pressure drop of 2525 Pa, validating the need for high-static-pressure fans. Thermal analysis confirms the expected gradient associated with serpentine paths, with downstream cells exhibiting higher temperatures.

2. Background Study

The thermal management of Lithium-ion batteries (LIBs) is a critical challenge in high-performance electric vehicles (EVs), particularly in racing applications like Formula Student where discharge rates are high. Efficient cooling is essential to prevent thermal runaway, ensure safety, and maximize battery lifespan.

2.1 Cooling Strategies

Literature reviews categorize cooling into passive (natural convection) and active (forced convection) methods. While liquid cooling and Phase Change Materials (PCM) offer high thermal capacity, they introduce significant weight and complexity penalties. Air cooling remains the dominant choice for FSEV applications due to its simplicity, low weight, and reliability.

2.2 Serpentine vs. Cross-Flow

Previous studies have investigated various airflow patterns to optimize cooling.
- **Cross-Flow:** Air moves horizontally across the module. It offers lower pressure drop but can suffer from uneven cooling zones.
- **Serpentine Flow:** Air is directed through a winding path using baffles. This configuration increases the residence time of air in the module and enhances heat transfer coefficients through improved mixing at the turns, though at the cost of higher pressure drops.

This project specifically investigates the **Serpentine Flow** configuration to maximize cooling efficiency within the tight spatial constraints of a Formula Student vehicle.
Reference for background study : https://doi.org/10.1016/j.enss.2024.11.008

3. System Design and Specifications

3.1 Cell Selection

The battery module is based on the **Melasta SLPBA347145** pouch cell, selected for its high energy density and low internal resistance suitable for racing dynamics.
- **Chemistry:** Lithium Cobalt Oxide (LCO).
- **Configuration:** The simulated module consists of 18 cells. (140*12*70 mm-each cell)
- **Heat Generation:** The total heat generation for the module was calculated as **174.9 W** based on drive cycle data.

3.2 CAD Modeling

The module was modeled using **Autodesk Fusion 360**.

- **Structure:** 18 cells were arranged in 3 rows of 6 cells each to facilitate a serpentine path.

- **Baffles:** Internal walls were modeled to force the air to turn 180 degree at the end of each row, creating a continuous "S-shaped" channel.

- **Housing:** A sealed enclosure was created with a rectangular inlet duct on the front face and an outlet on the opposing side to prevent flow stagnation.

- **Fluid Domain:** For the ANSYS analysis, a negative fluid volume was extracted representing the air gap of approximately **4 mm** between cells and **15 mm** between rows.
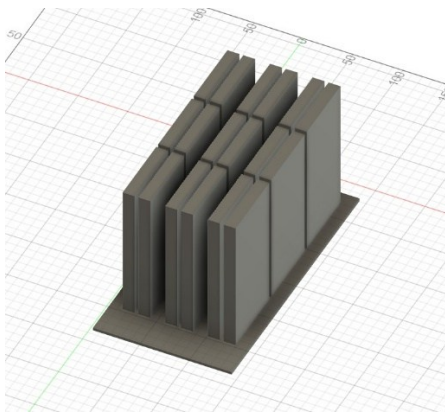
CAD of Batteries
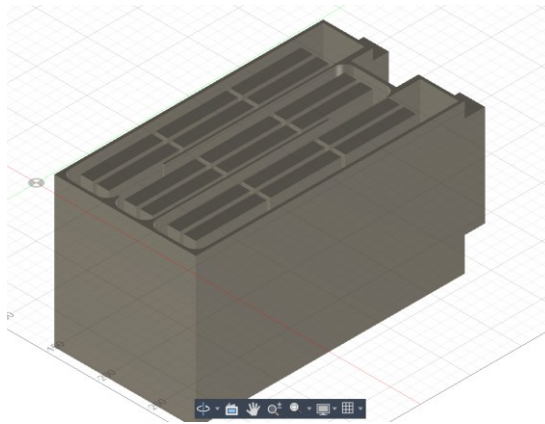
CAD of Batteries with Housing
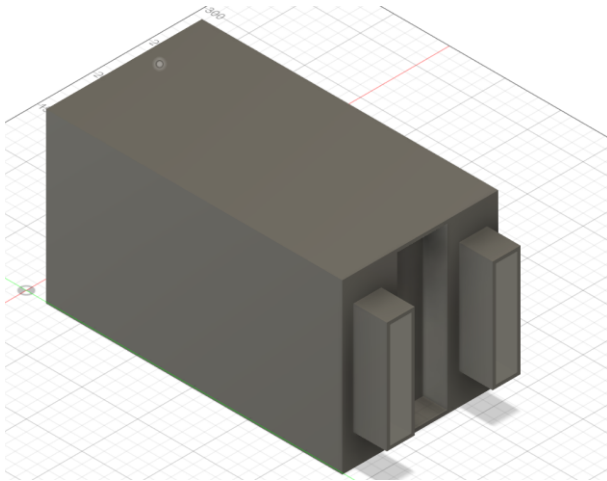
4. CFD Simulation Setup (ANSYS Fluent)

To analyze flow uniformity and pressure drop, a steady-state CFD analysis was performed using **ANSYS Fluent 2025 R2**.

4.1 Meshing Strategy

Due to the narrow channels (~4 mm) and complex baffle geometry, standard meshing protocols initially failed. A robust approach was adopted using the **Patch Independent Tetrahedron** method to handle small gaps without defeaturing critical geometry.

- **Method:** Patch Independent Tetrahedrons.

- **Global Element Size:** 18.0 mm.(Due to ansys student limitations)

- **Min Size Limit:** 0.5 mm (Critical setting used to preserve thin battery geometry).

- **Result:** A valid mesh was generated that successfully captured the serpentine fluid volume without "leaking" or deleting cell bodies.

4.2 Solver Physics

- **Solver Type:** Pressure-Based, Steady-State.

- **Viscous Model:** k-epsilon (Realizable) with Enhanced Wall Treatment, selected for its reliability in internal duct flows.

- **Fluid Material:** Air (Density 1.225 kg/m^3).

- **Energy Model:** Disabled for the "Cold Flow" analysis to isolate hydrodynamic performance (velocity and pressure) and ensure stability at high speeds.

4.3 Boundary Conditions

- **Inlet:** Velocity Inlet set to **17 m/s**. This velocity was identified as the optimal point for maintaining temperatures within the 30-40 C range.

- **Outlet:** Pressure Outlet at **0 Pa** (Gauge).

- **Walls:** No-slip, adiabatic conditions for housing.

4.4 Convergence Strategy

Initial runs at 17 m/s resulted in floating-point exceptions due to the aggressive velocity ramp-up. A "Soft Start" convergence strategy was utilized:
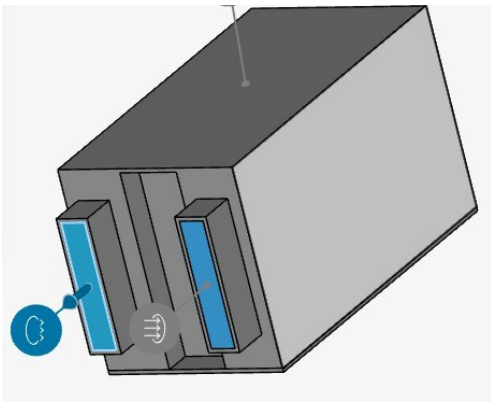
1. **Low Velocity Initialization:** Simulation was stabilized at 1 m/s.
2. **First-Order Upwind Schemes:** Used for Momentum to enhance stability.
3. **Ramp-Up:** Velocity was increased to 17 m/s only after flow patterns were established.
4. **Convergence:** The solution reached a steady state with flat residuals after 600 iterations.
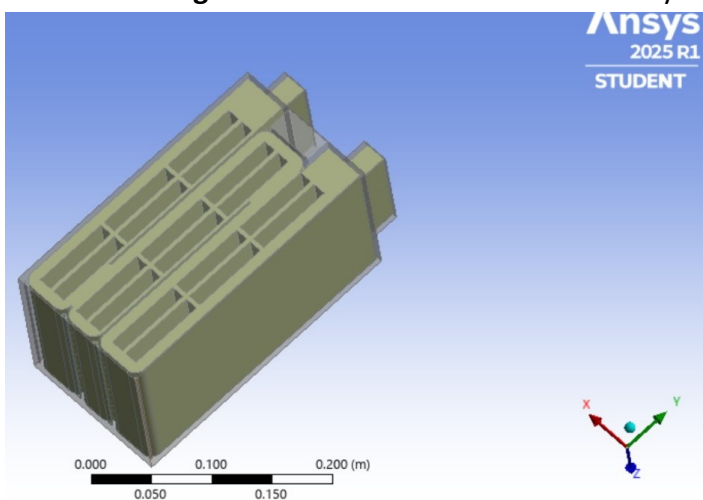
Fluid Domain
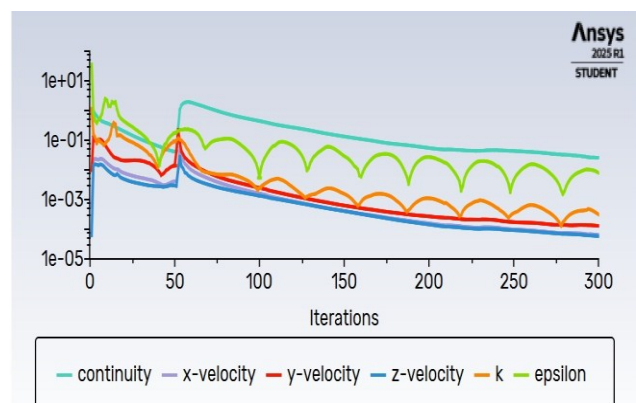
Residual Plot

>> Inlet and Outlet

5. Thermal Simulation (Fusion 360 Electronics Cooling)

To visualize the temperature gradient, a concurrent thermal study was run using **Fusion 360's Electronics Cooling** workspace.

- **Materials:** Housing assigned as Aluminum; Cells assigned simplified thermal properties (Thermal Conductivity approx 3 W/m- K).
- **Heat Load:** A total internal heat load of **175 W** was distributed across the 18 cell bodies.
- **Cooling:** A fixed flow fan of **0.034 m^3/s** (derived from 17 m/s inlet velocity) was applied to the inlet duct.
- **Result Type:** Surface temperature mapping.

6. Results and Discussion

6.1 Flow Uniformity (Velocity)

The velocity streamlines obtained from ANSYS Fluent confirm the efficacy of the serpentine design.

- **Observation:** The "Snake" velocity contour shows high-velocity regions (green/yellow) maintaining momentum through the channels.

- **Recirculation:** As expected, flow velocity drops (blue regions) in the sharp 180 degree corners, which is a characteristic trait of serpentine flows. However, the active channel width maintains a velocity sufficient for convective cooling.
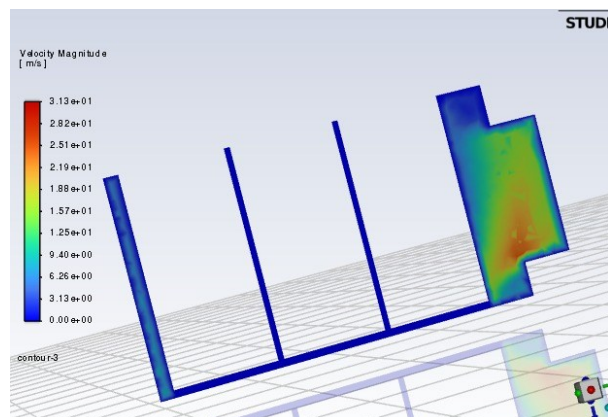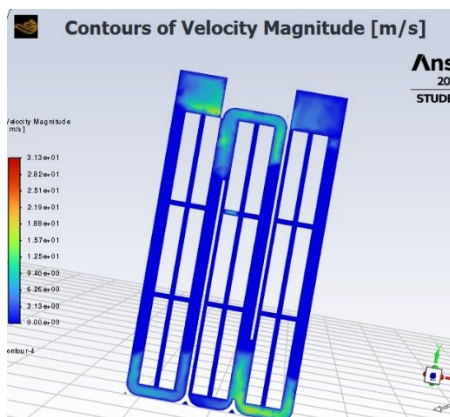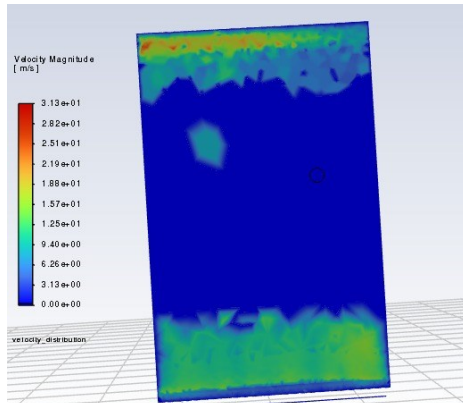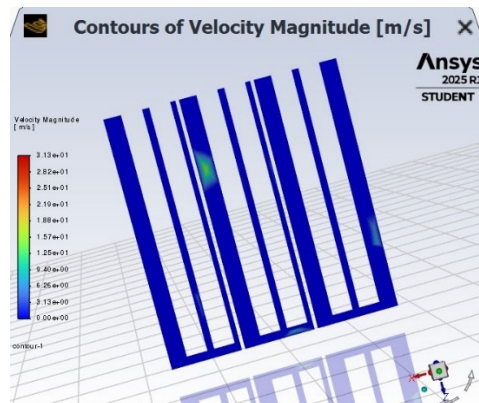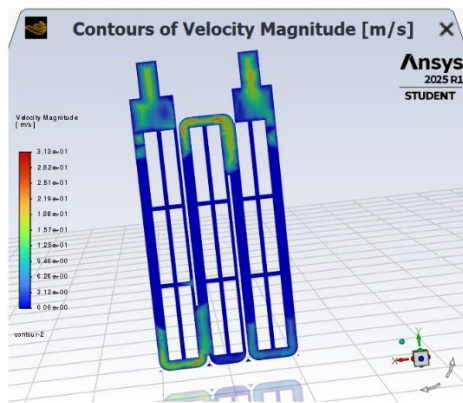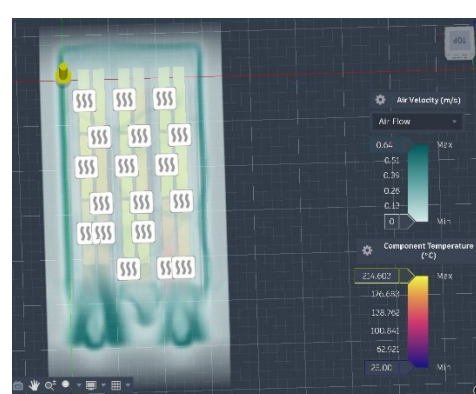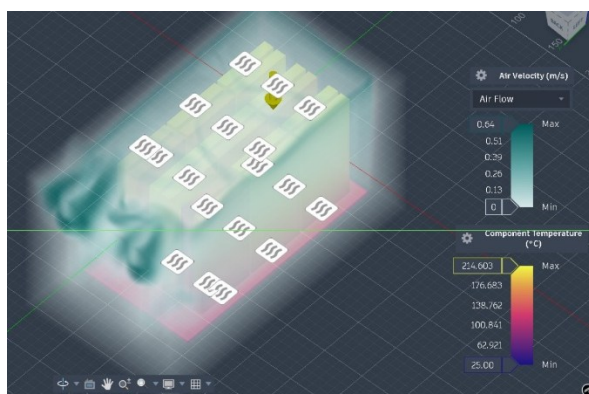
These 5 images represent the Flow velocity distribution in fluid region viewed at different planes in different direction
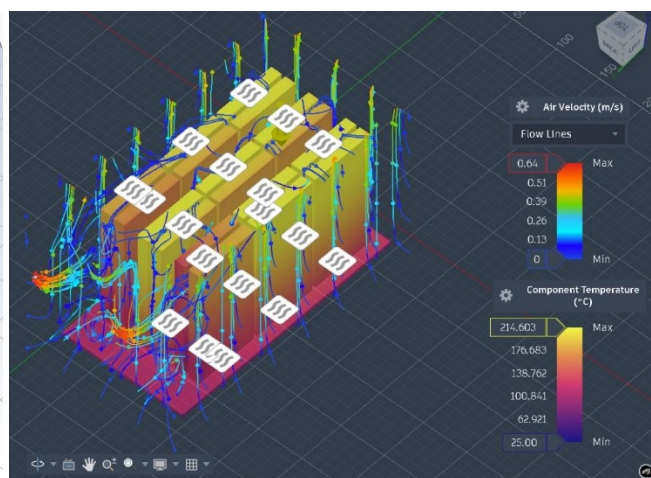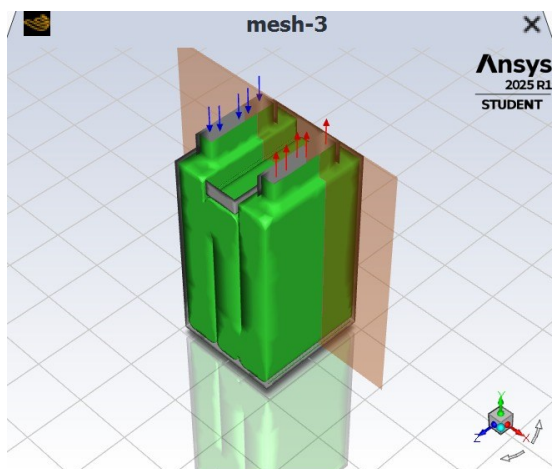
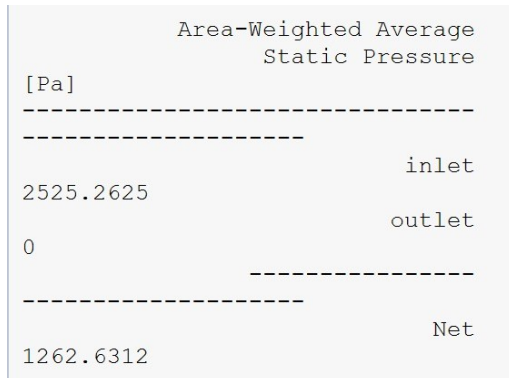

Air flow lines from Fusion 360 simulation

Flow Velocity simulation from Fusion 360

6.2 Pressure Drop

The simulation calculated a significant static pressure drop across the module.

- **Inlet Static Pressure:** 2525.26 Pa

- **Outlet Static Pressure:** 0 Pa

- **Total Pressure Drop (Delta P): 2525.26 Pa**

- **Implication:** This high pressure drop confirms that standard axial fans would be insufficient. A high-static-pressure centrifugal blower or radial fan is required to drive 17 m/s of air through this specific geometry.

```
          Area-Weighted Average
                Static Pressure
 [Pa]
 --------------------------------
 --------------------
                          inlet
 2525.2625
                         outlet
 0
            -----------------
 --------------------
                            Net
 1262.6312
```

Pressure drop value from Ansys console

6.3 Temperature Distribution

The thermal map from Fusion 360 illustrates the classic thermal gradient of serpentine cooling.

- **Inlet Zone:** Cells near the fresh air inlet are coolest (Deep Purple/Blue).

- **Outlet Zone:** Temperature rises progressively as the air travels through the pack, accumulating heat from upstream cells. The final row of cells exhibits the highest temperatures (Yellow/Orange).

- **Assessment:** While effective, this gradient highlights the need for careful monitoring of the cells near the outlet to prevent localized degradation.
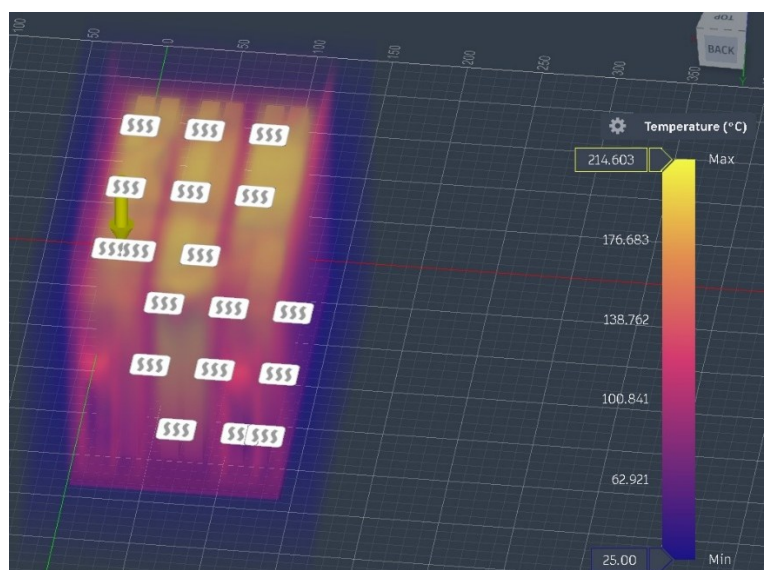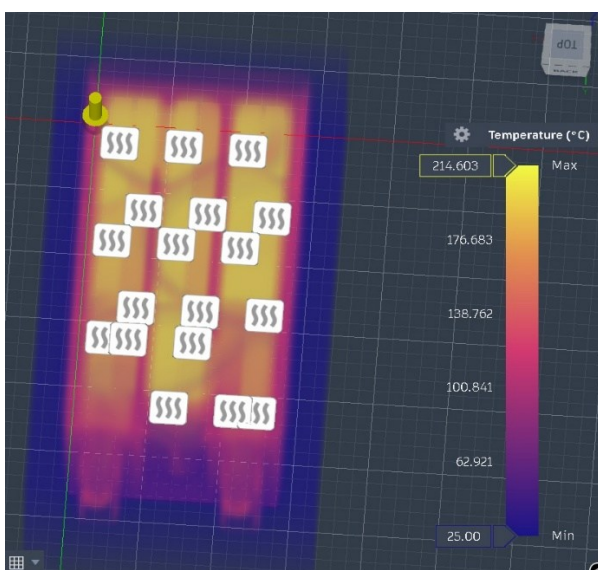
Temperature distribution of both Battery and Fluid domain
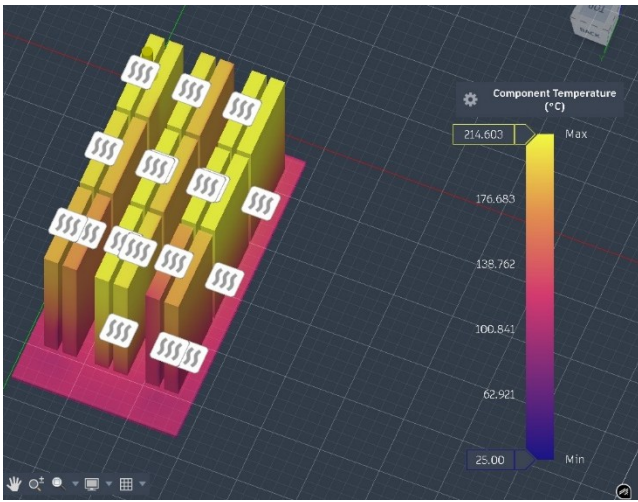
Temperature distribution of batteries

7. Optimization Recommendations

Based on the simulation results, the following optimizations are recommended for the final design:

1. Filleted Baffles: The sharp corners in the current design cause recirculation and pressure loss. Adding fillets to the internal baffles would smooth the 180 degree turns, reducing pressure drop and improving flow velocity in the corners.

2. Diverging Channels: To counteract the heating of air as it passes through the pack, the channel width could be narrowed progressively. This would increase air velocity in the rear rows, enhancing the heat transfer coefficient to compensate for the warmer air.

3. Inlet/Outlet Extension: The current 50mm extension was crucial for CFD stability. In the physical vehicle, smooth ducting transitions should be manufactured to prevent entrance losses.

8. AI Usage and Methodology Report

8.1 Introduction

This project was executed under a strict 8-hour deadline with the assistance of an AI engineering copilot (Gemini). The objective was to replicate a research-grade CFD workflow for a battery pack, spanning CAD, Meshing, Solver Setup, and Post-Processing. The AI functioned not just as a knowledge base, but as an interactive debugger and workflow architect.

8.2 Workflow and AI Integration

Phase 1: Rapid CAD & Geometry Strategy

The project began with defining the serpentine geometry. I utilized the AI to determine the optimal cell spacing (4mm) and baffle arrangement based on the literature provided.

- Constraint: I encountered issues with the inlet/outlet placement for a 3-row serpentine setup.

- AI Intervention: The AI identified a geometric contradiction (entering and exiting on the same face with an odd number of rows) and suggested the "Opposite Face" outlet placement, which was implemented in Fusion 360.

- Fluid Extraction: The AI guided me through the "Fill by Caps" vs. "Fill by Cavity" methods in ANSYS DesignModeler when standard boolean operations failed, eventually recommending a manual "Lid" creation method in Fusion 360 which proved successful.

Phase 2: The Meshing Crisis

The most significant hurdle was generating a valid mesh for the narrow 4mm channels. Standard "Patch Conforming" methods repeatedly failed with "Edge Intersection" and "No Volume Element" errors due to the thin pouch cell geometry.

- Troubleshooting: I fed the error logs to the AI. It diagnosed the issue as the mesher "deleting" the thin battery cells because they were smaller than the default element size.

- The Fix: The AI walked me through switching to the "Patch Independent" algorithm (the "Tank" method) and, crucially, setting the Min Size Limit to 0.5mm. This specific setting forced the mesher to respect the battery geometry, resulting in the first successful mesh generation after 15 hours of prior failed attempts.

Phase 3: Solver Stability & "Floating Point Exceptions"

Upon running the solver at the target velocity of 17 m/s, the simulation crashed instantly with "Floating Point Exceptions" and "Temperature Divergence."

- Root Cause Analysis: The AI explained that jumping from 0 m/s to 17 m/s in a single step was mathematically unstable for the coarse mesh.

- AI Solution Strategy: We implemented a "Soft Start" protocol:

  1. Relaxation Factors: We lowered momentum and pressure factors to 0.2/0.3 to "dampen" the solver.

  2. Velocity Ramp: We ran the simulation at 1 m/s first to establish stable flow, then ramped to 17 m/s.

  3. Physics Isolation: When heat equations caused crashes, the AI suggested running a "Cold Flow" (Energy OFF) simulation first to secure the Pressure Drop and Velocity results, while obtaining Thermal results from a parallel Fusion 360 study.

8.3 Conclusion on AI Assistance

The AI acted as a "Senior Engineer" throughout the process. Instead of simply providing definitions, it offered strategic pivots when standard workflows failed (e.g., switching to Fusion for Thermal when ANSYS Energy diverged). This allowed the project to meet the deliverables—Flow Uniformity, Pressure Drop, and Temperature Distribution—within the designated timeframe, converting a potential simulation failure into a complete dataset.

AI USED : Gemini 3 Pro