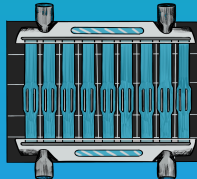


Thermofluids Group Coursework:

SUV & Battery Cooling Pipeline Design



DE2 Thermofluids
Group 23

Cici Song, Yuxuan Zhou, Simon Xia, Joe Chen
CID: 02563502, 02569052, 02580218, 02570369

SUV Design

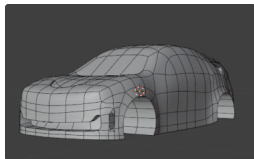
- Project Summary ●
- Printed Model ●
- Wind tunnel testing ●
- Result Comparison ●
- Design Optimisation ●

Pipeline Design

- Design Summary ●
- Design Rationale & Conceptual Development ●
- Modeling ●
- Pressure Loss Hand Calculation ●
- 1-D Heat transfer hand-calculation ●
- CFD Setup ●
- Meshing ●
- CFD Output ●
- Comparison & Discussion of Results ●
- Brief Discussion of Results ●
- Future Recommendation ●
- Team & Project Management ●

Project Summary

Selected Concept: SUV Design



Design purpose

The SUV concept targets better **fuel efficiency** by **controlling separation** and **wake development**.

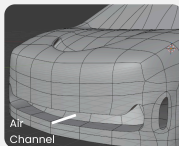
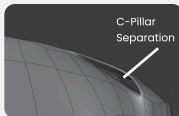
Key Statistics

- Frontal Area for model in wind tunnel: **0.00833m²**
- Cd from wind tunnel testing: **0.31**
- t1 Frontal Area: **2.41m²**
- Length : **4.607m**
- Width : **1.92m**
- CL=**0.206**



Current Design Features

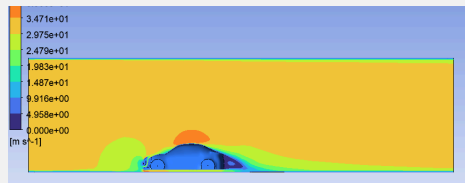
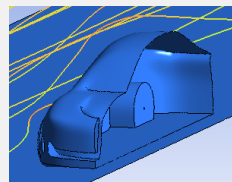
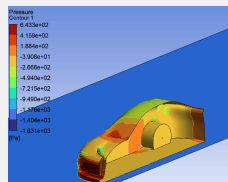
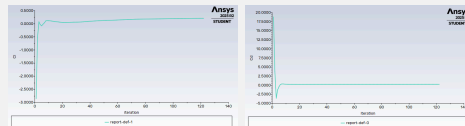
- Front air-curtain:** Directs **high-energy air** around the **front wheel** region to weaken tyre-induced **vortices** and reduce **local separation**.
- Smooth body-side and roof surfaces:** Minimises **abrupt curvature** changes to help maintain **attached flow** and delay **separation** toward the rear.
- SF90-inspired front channelling:** Creates a **controlled flow path** from the **front region** to keep surface flow **organised** and reduce external flow disturbance.
- i8-inspired C-pillar separation:** Uses a defined geometric **split** to organise side flow, mitigating **rear-side separation** and yaw-induced pressure asymmetry.
- Rear-end separation management:** Shapes the rear to **anchor separation** more predictably, aiming to reduce **recirculation volume** and wake thickness.



CFD Results

Cd: 0.286 Cl: 0.24

The body shows **low aerodynamic drag** for an SUV-style geometry, suggesting **efficient overall shaping** and **reasonable rear pressure recovery**. The **Cl=0.24** value is moderately **positive**, indicating net lift that could **slightly reduce** high-speed stability.

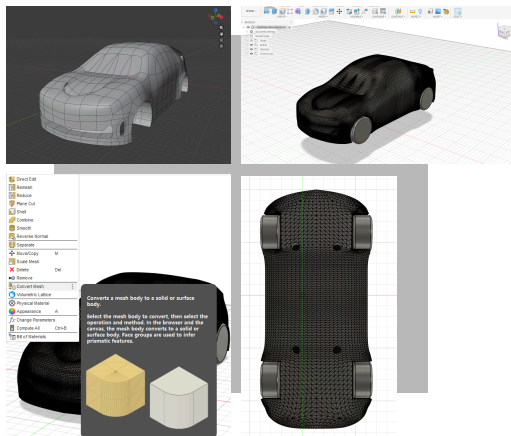


Printed Model

Purpose

To present the digital preparation, print setup, and final physical prototype of the model.

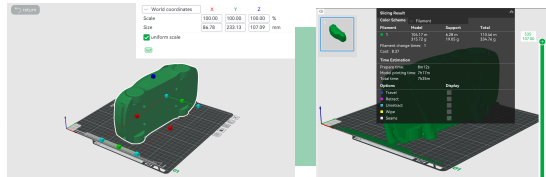
Fusion Setup



The vehicle geometry was initially modelled in **Blender** and subsequently imported into **Fusion 360** for further preparation. In Fusion, the mesh model was converted into a solid body to enable downstream CFD analysis and manufacturability adjustments.

To prepare the model for wind tunnel testing, **4 mounting holes for M8 screws** were integrated into the base. This allows secure fixation during testing. The final geometry was then exported as an **STL file** for 3D printing.

Bamboo Setup



- The model was scaled to meet the constraints of the wind tunnel testing setup, resulting in final dimensions of **233 mm (length)**, **86.78 mm (height)**, and **107 mm (width)**.
- The orientation and printing parameters were carefully optimised in **Bambu Studio** to minimise printing time, reduce support structures, and improve surface quality.

This configuration achieved a balance between **print efficiency** and **geometric accuracy**, with a total printing time of approximately **7 hours 25 minutes**.

Printed Model



The model was fabricated using a **Bambu X1 Carbon 3D printer** with **PLA material**. The final printed prototype demonstrates good geometric fidelity to the original CAD model, with smooth surface finishes and sufficient structural rigidity for wind tunnel testing. The physical model enables experimental validation of the aerodynamic design and provides a tangible representation of the final concept.

Wind tunnel testing

Purpose To present the wind tunnel setup, flow observations, and drag results.

Wind tunnel testing is used to evaluate the **aerodynamic performance** of the SUV model under controlled **airflow conditions** and support further design optimisation.

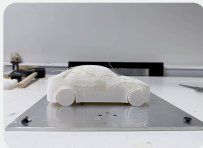
Setup

- a. Model Scale:** The model is a **1:19.6** scale of the real car that each linear dimension is reduced by a factor of **19.6**. The frontal area is **0.00833 m²** after scaling down.
- b. Airspeed:** The airspeed settings are **10 m/s, 20 m/s and 30 m/s** to simulate the different real-world cruising speed conditions of the SUV car.
- c. Reynolds number**
 - **10 m/s:** $Re \approx 5.95 \times 10^4$
 - **20 m/s:** $Re \approx 1.19 \times 10^5$
 - **30 m/s:** $Re \approx 1.78 \times 10^5$

As the tunnel speed increases from **10 to 30 m/s**, the Reynolds number rises, so inertial effects become more dominant than **viscosity**. Cd decreases with speed, which suggests a **Reynolds effect**: lower Re promotes earlier separation and a larger wake, while higher Re tends to delay separation and stabilise the flow, reducing Cd.

d. Model installation

- **Tufts** were attached to the model surface to **visualize** near-surface flow direction and detect separation.
- The model was secured to the mounting plate using **four M8 screws**.
- The **metal mounting plate** was fastened to the wind-tunnel floor with screws to ensure a rigid, vibration-free setup.



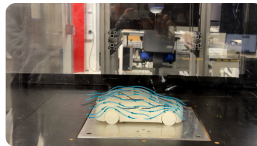
Testing Conditions

Wind tunnel testing was conducted at three inlet air speeds of **10, 20, and 30 m/s** under standard atmospheric pressure (**101 kPa**). The reference frontal area was fixed at **A=0.008 m²** for all cases.

Using multiple speeds captures performance across low to higher tunnel conditions and helps check repeatability. As velocity **increases**, the dynamic pressure **increases** approximately with **V²**, so the measured **drag and lift forces** are expected to **rise** significantly with speed even if the geometry is unchanged.

To enable **fair comparison** between the three speeds, results are reported as non-dimensional coefficients using **C=F/(qA)**. Keeping constant removes the direct **V²** scaling, so changes in **Cd** or **Cl** more clearly reflect aerodynamic behaviour rather than test speed.

Conducting Wind Tunnel Test



The model was placed in the wind tunnel, the freestream velocity was increased from **0 m/s**, and steady test points were recorded at **10 m/s, 20 m/s, and 30 m/s**. Tuft visualisation was used to identify regions of **unsteady separated flow**. Tufts that fluttered rapidly or reversed direction indicated locally turbulent and chaotic flow near the surface.



Across the tested freestream velocities of **10 m/s, 20 m/s, and 30 m/s**, the tufts over the front and roof remained largely aligned with the incoming flow, while the tufts at the rear were the only ones to show persistent **unsteady motion**. As the **airspeed** increased the **rear tufts** oscillated faster and with larger **amplitude**, indicating progressively stronger **rear end flow separation** and a more energetic **turbulent wake** with recirculation at the base. This consistent rear only response identifies the tail region as the dominant separation site and the main contributor to pressure drag in the wind tunnel observations.

Drag Coefficient

Frontal Area A: 0.00833 m²

Dynamic Pressure Q: 4.451 Pa in 10m/s, 18.213 Pa in 20m/s, 59.2205 Pa in 30m/s

D & L: Drag force and Lift Force

$C_d = \frac{D}{qA}$ When air speed is **10m/s**, the drag force is around **0.19 to 0.20N**, with an area of **0.00833**, the Cd is calculated to be **0.37**.

When air speed is **20m/s**, the drag force is around **0.68 to 0.70N**, with an area of **0.00833**, the Cd is calculated to be **0.32**.

$C_l = \frac{L}{qA}$ When air speed is **30m/s**, the drag force is around **1.53 to 1.54N**, with an area of **0.00833**, the Cd is calculated to be **0.31**.

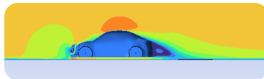
Result Comparison

Purpose

To compare the CFD and wind tunnel results, highlight key similarities and differences, and explain the main sources of discrepancy.

Comparison of parameters

Comparison	CFD Result	Wind Tunnel Result	Difference	Comment
Drag coefficient, Cd	0.286	0.31	7.8%	CFD predicts slightly lower drag
Lift coefficient, Cl	0.24	0.206	16.5%	CFD predicts more positive lift
Drag force at 10 m/s	0.158N	0.19–0.20 N	26.6%	CFD predicts significant lower drag
Drag force at 20 m/s	0.633N	0.68–0.70 N	7.42%	CFD predicts slightly lower drag
Drag force at 30 m/s	1.424N	1.53–1.54 N	7.44%	CFD predicts slightly lower drag



*The Drag force is calculated under few assumptions:

- Air density, $\rho=1.225 \text{ kg/m}^3$
- Frontal area = 0.00833 m^2
- Drag coefficient $C_d=0.31$

Flow feature Comparison

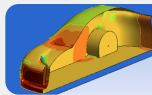
Similarities:

- Both methods indicate attached flow over the **front and roof**.
- Both identify the **rear** region as the **main area of flow separation**.
- Both suggest that a wake forms **behind** the vehicle, contributing to **drag**.

Differences:

- The **rear separation region** appears more **defined** and **continuous** in CFD, whereas in the wind tunnel it appears more **irregular** and **less structured**.
- The CFD flow pattern looks **smoother** and more **stable**, while the wind tunnel result shows more local **fluctuation** and unsteadiness due to real physical airflow.

Analysis of Discrepancies



This section outlines several possible reasons for the **discrepancy** between the two methods, ranked from **most to least** significant. Lower-ranked factors are expected to have a smaller effect on the overall difference.

1

Mesh Quality & Resolution

The rear body should reduce in **cross-section** more progressively instead of ending too abruptly. This would help the airflow stay attached for **longer, delay separation**, and reduce the size of the **wake** behind the vehicle.

2

Turbulence model limitations

The CFD simulation relies on a **turbulence model** to approximate **complex airflow** behaviour. However, turbulence models cannot perfectly **capture** all **real** flow phenomena, especially in regions of flow separation and wake formation behind the vehicle

3

Surface finish and manufacturing imperfections (3d printing defect)

The printing **geometry** and **quality** are limited with 3d printer in our workshop. These **imperfections** can disturb the **airflow** and slightly increase **drag**, whereas the CFD model usually assumes an ideal **smooth** surface, e.g. support not fully removed, roughness

4

Uncertainty in experiment

Despite the fact **three** experiment is down with **three** distinct wind speed. **Multiple experiment** is still needed to **average** and get the final result. Small errors in **force readings** or **airspeed control** can also contribute to discrepancies between measured and simulated values.

Design Optimisation

Purpose

To evaluate CFD accuracy, identify key aerodynamic weaknesses, and propose targeted design improvements.

Validation of CFD Prediction

7.42% Drag force difference

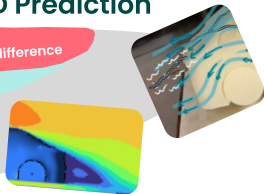
7.44%

Cd Difference

7.8%

CI Difference

16.5%



The wind tunnel test reinforced the CFD finding that the rear region is the main weakness. In the tuft test, the rear tufts showed **persistent oscillation** and **unstable motion**, consistent with the low-velocity recirculation region predicted in CFD. These results suggest that the CFD model correlated well with the real test, particularly for lift and drag force prediction

Challenges

1

Low-speed drag prediction was the weakest

At **10 m/s**, the drag force difference reached **26.6%**, much larger than the differences at **20 m/s** and **30 m/s**, indicating weaker experiment agreement under lower-speed conditions.

2

CFD produced a smoother flow field than reality

The CFD result shows a **cleaner** and more **stable** flow pattern, while the wind tunnel visualisation reveals **more local fluctuation and unsteadiness**

3

Coefficient error in lift prediction

16.5% difference in CI suggests that the CFD model was **less accurate** in predicting the vertical aerodynamic behaviour and slightly overpredicted lift compared with the wind tunnel result

Aerodynamic Weaknesses

1

Unstable rear structure

The **rear** is the main region where airflow detaches from the body, which largely reduces **aerodynamic efficiency**. The tufts near the rear oscillated more strongly than those on the **front and roof**, showing that the rear geometry does not control the wake effectively.



2

Wheelhouse disturbance

The design still suffers from **pressure drag**, while the CFD analysis shows **disturbed flow** near the wheelhouse. This region is generating turbulence, which adds **drag and disrupts** the side flow. Although the air curtain appears helpful, this area is not fully resolved.



3

Vertical aerodynamic behaviour

The body's vertical response is **less predictable** due to the difference in CI between **CFD** and **wind tunnel test**, which means it may not manage vertical forces with the same consistency as it manages overall drag



Specific design improvement

1

Introducing a gradual rear taper

The rear body should reduce in **cross-section** more **progressively** instead of ending too abruptly. This would help the airflow stay attached for **longer, delay separation**, and reduce the size of the wake behind the vehicle.

2

Reduce wheelhouse exposure

The **front wheel** opening should be better shielded. A cleaner **wheel arch edge** and tighter control of incoming side flow would reduce **turbulence generation** around the rotating wheel region

3

Strengthen the front air curtain

Air curtain should be **reshaped** to push air more **cleanly** around the front wheel. This would reduce **wheelhouse ingestion, local vortices, and side-flow disturbance**.

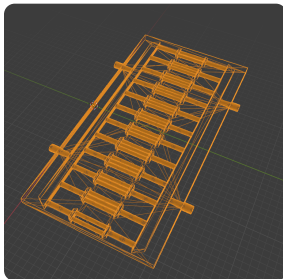
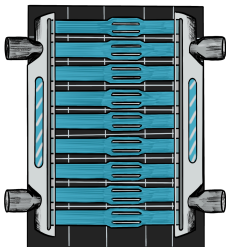
4

Smooth local surface discontinuities

small **geometric irregularities** may be contributing to extra disturbance in real flow. Any **abrupt edges, uneven transitions, sharp section changes**, or local surface breaks should be refined

Design Summary

This refined cold plate design uses a **parallel micro-channel layout** to improve **heat transfer** and **cooling uniformity**. A manifold area is included to **distribute and collect** coolant more evenly, helping to reduce flow maldistribution between channels. The use of **double inlets** and **double outlets** further shortens the flow path and supports more balanced coolant flow across the plate, improving overall thermal performance.



Key Design Features

- Parallel micro-channels** increase the effective **heat transfer area** and strengthen **convective cooling** between the coolant and the cold plate. This helps improve local heat removal and supports lower battery surface temperature.
- Manifold regions** are introduced to distribute and collect coolant more evenly across the parallel channels. This helps reduce flow maldistribution and allows the cooling performance to be more uniform across the plate.
- Double inlets and double outlets** shorten the overall coolant flow path and promote a more balanced flow pattern from both sides of the plate. This helps improve thermal uniformity while avoiding an excessive pressure-drop increase.

Intended Application

- Designed for EV battery pack thermal management
- Used to remove heat generated during battery charging and discharging
- Aims to reduce hotspots, Tmax, and temperature non-uniformity
- Supports safe, stable, and efficient battery operation
- Helps improve battery lifetime and thermal reliability

Calculation Result Summary

Hand- Calculation:

Calculated pressure loss: 1.399Pa

T_Max : 91.19 °C

$$\Delta P_{\text{long}} = f \left(\frac{L}{D_h} \right) \left(\frac{\rho V^3}{2} \right)$$

$$\Delta P_{\text{long}} = 0.298 \left(\frac{0.07}{0.006} \right) \left(\frac{997 \times \frac{0.02395^3}{2}}{2} \right) = 0.746 \text{ Pa}$$

$$\Delta P_{\text{bend}} = 0.298 \left(\frac{0.032}{0.006} \right) \left(\frac{997 \times \frac{0.02395^3}{2}}{2} \right) = 0.395 \text{ Pa}$$

$$\Delta P_{\text{entrance}} = 0.670 \left(\frac{0.023}{0.006} \right) \left(\frac{997 \times \frac{0.01426^3}{2}}{2} \right) = 0.258 \text{ Pa}$$

$$\Delta P_{\text{total, straight}} = \Delta P_{\text{long}} + \Delta P_{\text{bend}} + \Delta P_{\text{entrance}}$$

$$\Delta P_{\text{total, straight}} = 0.746 + 0.258 + 0.395 = 1.399 \text{ Pa}$$

$$V_b = 0.006174 \text{ m}^3/\text{s}$$

$$Q = 740.88 \text{ W}$$

$$T_{\text{exit}} = 39.77^\circ\text{C}$$

$$dT_c = 14.77^\circ\text{C}$$

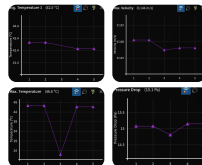
$$q'' = \frac{83999 \text{ W}}{\text{m}^2}$$

$$T_{\text{max}} = 91.19^\circ\text{C}$$

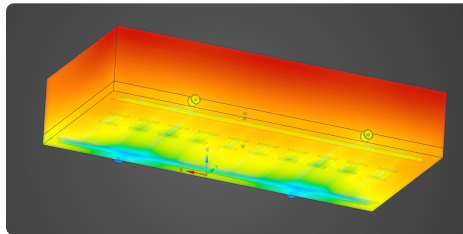
CFD Estimation:

Calculated pressure loss: 15.1 Pa

T_Max : 46.6 °C



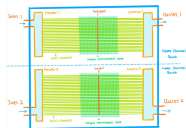
CFD Result



Design Rationale & Conceptual Development

Benchmarking of Existing Solution

Design 1: Parallel Microchannels



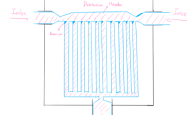
Pros

- Parallel channel reduces overall **flow velocity**, enabling controllable **pressure drop**.
- Hotspot enhancement reduces T_{max} .
- Fin** features increase the effective heat-transfer area, reducing ΔT .

Cons

- Increased **geometric complexity** raises manufacturing difficulty.
- Thinner hotspot zone may cause risk of local high Δp .

Design 2: Multi-inlet parallel channels



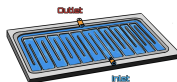
Pros

- Improve **coverage** and temperature uniformity.
- Dual inlets improve **symmetry** and **header pressure** balance.
- Straight channels** are relatively easy to machine and seal.

Cons

- Restrictors increase total **pressure drop** and **pumping power**.
- Narrow restrictors can **clog**.
- Dual-inlet** adds **complexity** and potential leak points.
- Restrictors increase Δp and are clogging-sensitive.

Design 3: Serpentine-parallel channels



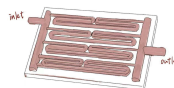
Pros

- Repeated channels improve **temperature uniformity**.
- Hybrid** layout reduces **pressure drops** compared to full serpentine.
- Single loop** structure reduces manifold surfaces.

Cons

- Risk of **uneven** velocity distribution across parallel channels.
- flow **decelerates** near turning.
- the coolant removes heat less **aggressively**.
- More turns in the channel mean more resistance to flow.

Design 4: Serpentine-parallel



Pros

- Low system complexity.
- Broad channel coverage improves **utility**.
- Lower restriction reduces **pressure drop** and **pumping demand**.
- Easier to **manufacture** and **integrate**.

Cons

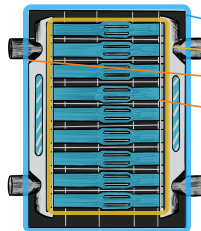
- Parallel branches** may not distribute flow evenly.
- Manifold pressure** changes can cause branch imbalance.
- Long flow paths increase overall **pressure loss**.

Concept Selection & Refined Model

Design 1 is selected and refined to a **parallel** channel configuration with **integrated fin structures** for **hotspot enhancement**, effectively reducing both ΔT and T_{max} while maintaining a manageable pressure drop (Δp).

Several refinements are incorporated into Design 1:

- Tapered transitions** are introduced to **minimise** minor **hydraulic losses** and prevent unnecessary pressure-drop increases at geometric transitions.
- Restrictors** are implemented at the entrance of each parallel channel to intentionally increase local **hydraulic resistance**, thereby improving flow distribution uniformity across the network.
- The original **dual-bank** configuration is consolidated into a single bank to improve effective **heat-transfer** area utilisation and reduce **manufacturing** and **sealing complexity**.
- The **micro-channel architecture** is simplified into multiple structured parallel channels to reduce geometric complexity while preserving thermal performance.



- Cold Plate Size: **420*210*70 mm**
- Active heat transfer area: **400 × 190 mm**
- Inlet manifold width: **12 mm**
- Main channels (parallel): **12 channels**
Single channel length: **≈ 190 mm**
Port outer diameter : **Ø10–12 mm**
Port inner diameter (ID): **Ø6 mm**
- Outlet manifold width: **12 mm**

Modeling

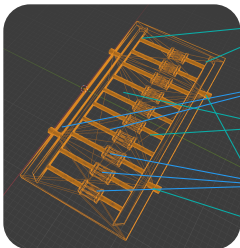
Purpose

To present the modelling overview, key design features, and flow-control strategies of the pipeline concept.

Overview

A model of the pipeline design is constructed to a 3D model using **Fusion 360**. Features including parallel channels, tapering, and micro-channels. Restrictors are not included due to the complexity of modelling.

Perspective View



Manifold

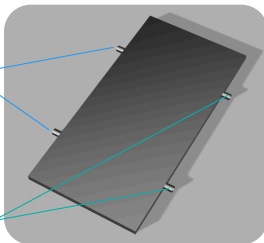
Inlets

Parallel Channel

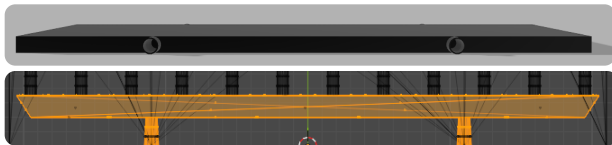
Micro channels

Micro channels

Exterior View

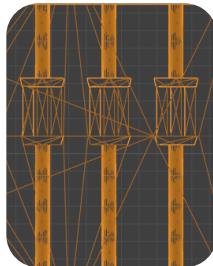


Inlet & manifold

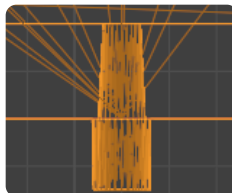


The manifold zone acts as a **transition** between the **inlet/outlet ports** and the internal channel network. It conditions the incoming flow by **reducing** velocity peaks and equalising static pressure before the flow splits, which helps minimise flow maldistribution across channels. Smooth area changes and radiused turns are used to avoid local separation and keep the pressure-drop penalty acceptable, balancing uniformity with manufacturability.

Design Information



Parallel microchannels use many identical **small channels** in parallel to increase **wetted area** and boost heat transfer, spreading cooling more **evenly** across the plate. The **trade-off** is higher **flow resistance**, so good **manifold** and **tapering** design is needed to keep distribution uniform and pressure drop acceptable.



Tapering is applied in the manifold transition to gradually change the **flow area** and local **hydraulic resistance**. By narrowing or widening the passage progressively, it controls the **pressure gradient** along the distribution path so each branch "sees" a more similar driving pressure, which improves flow uniformity and reduces maldistribution.

A **smooth taper** also avoids **sudden expansions**, helping to limit separation losses and keeping the **pressure-drop** increase modest.

Pressure Loss Hand Calculation

Key Parameters & Dimensions

Cold plate

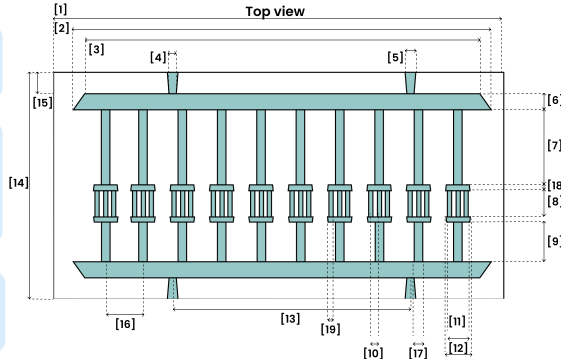
- [1]: length of active cold plate, **420mm**
- [14]: width of active cold plate, **210mm**

Manifold

- [2]: lower base of trapezoidal manifold, **410mm**
- [3]: upper base of trapezoidal manifold, **390mm**
- [6]: height of trapezoidal manifold, **15mm**
- [*]: thickness of manifold, **13mm**

Inlet&Outlet

- [4]: inner inlet diameter, **8mm**
- [5]: outer inlet diameter, **10mm**
- [15]: inlet tapering length, **20mm**
- [13]: distance between inlets, **220mm**



Parallel channel

- [7]: longer channel length, **70mm**
- [9]: shorter channel length, **37mm**
- [17]: channel diameter, **8mm**
- [16]: distance between channels, **38mm**

Micro manifold

- [11]: upper base of trapezoidal micro manifold, **21mm**
- [12]: lower base of trapezoidal micro manifold, **24mm**
- [18]: height of trapezoidal micro manifold, **5mm**
- [*]: thickness of micro manifold, **5mm**

Micro channel

- [8]: length of micro channel, **23mm**
- [19]: micro channel diameter, **6mm**
- [10]: distance between micro channels, **8mm**

Calculations: Pressure Loss ($\Delta p = 1.390 \text{ Pa}$)

Assumption and condition:

- Inlet mass flow rate: **12 g/s**
- Coolant: **water**
- Coolant initial temperature: **25 °C**
- Outlet boundary condition: pressure outlet fixed at **0 Pa**

1. Convert mass flow rate to volume flow rate

$$Q_{\text{ref}} = \frac{m}{\rho}$$

$$Q_{\text{ref}} = \frac{0.012}{997} = 1.204 \times 10^{-5} \frac{\text{m}^3}{\text{s}}$$

So the total volumetric flow rate is:

$$Q_{\text{ref}} = 1.204 \times 10^{-5} \frac{\text{m}^3}{\text{s}}$$

2. Flow rate in one long channel branch and one micro channel

$$Q_{\text{branch}} = \frac{Q_{\text{ref}}}{10}$$

$$Q_{\text{branch}} = 1.204 \times \frac{10^{-5}}{10} = 1.204 \times 10^{-6} \frac{\text{m}^3}{\text{s}}$$

$$Q_{\text{micro}} = \frac{Q_{\text{branch}}}{3}$$

$$Q_{\text{micro}} = 1.204 \times \frac{10^{-6}}{3} = 4.013 \times 10^{-7} \frac{\text{m}^3}{\text{s}}$$

3. Cross-sectional area

$$A_{\text{main}} = \frac{\pi D_{\text{main}}^2}{4} = \frac{\pi (0.008)^2}{4} = 5.027 \times 10^{-6} \text{ m}^2$$

$$A_{\text{micro}} = \frac{\pi D_{\text{micro}}^2}{4} = \frac{\pi (0.006)^2}{4} = 2.827 \times 10^{-5} \text{ m}^2$$

4. Velocity in each section

$$V_{\text{long}} = \frac{Q_{\text{branch}}}{A_{\text{main}}} = 1.204 \times \frac{10^{-6}}{5.027 \times 10^{-6}} = 0.02395 \frac{\text{m}}{\text{s}}$$

$$V_{\text{short}} = V_{\text{long}} = 0.02395 \frac{\text{m}}{\text{s}}$$

$$V_{\text{micro}} = \frac{Q_{\text{micro}}}{A_{\text{micro}}} = 4.013 \times \frac{10^{-7}}{2.827 \times 10^{-5}} = 0.01420 \frac{\text{m}}{\text{s}}$$

5. Reynolds number

$$Re = \frac{\rho V D}{\mu}$$

$$Re_{\text{long}} = \frac{997 \times 0.02395 \times 0.008}{0.00089} = 214.7$$

$$Re_{\text{short}} = \frac{997 \times 0.02395 \times 0.008}{0.00089} = 214.7$$

$$Re_{\text{micro}} = \frac{997 \times 0.01420 \times 0.006}{0.00089} = 95.5$$

Since all Reynolds numbers are less than 2300, the flow is laminar.

6. Friction factor

$$f = \frac{64}{Re}$$

$$f_{\text{long}} = \frac{64}{214.7} = 0.298$$

$$f_{\text{short}} = \frac{64}{214.7} = 0.298$$

$$f_{\text{micro}} = \frac{64}{95.5} = 0.670$$

7. Total Pressure drop

$$\Delta P_{\text{long}} = f \left(\frac{L}{D} \right) \left(\frac{\rho V^2}{2} \right)$$

$$\Delta P_{\text{long}} = 0.298 \left(\frac{0.07}{0.008} \right) \left(\frac{997 \times 0.02395^2}{2} \right) = 0.746 \text{ Pa}$$

$$\Delta P_{\text{short}} = 0.298 \left(\frac{0.037}{0.008} \right) \left(\frac{997 \times 0.02395^2}{2} \right) = 0.395 \text{ Pa}$$

$$\Delta P_{\text{micro}} = 0.670 \left(\frac{0.023}{0.006} \right) \left(\frac{997 \times 0.01420^2}{2} \right) = 0.258 \text{ Pa}$$

$$\Delta P_{\text{total,weight}} = \Delta P_{\text{long}} + \Delta P_{\text{micro}} + \Delta P_{\text{short}}$$

$$\Delta P_{\text{total,weight}} = 0.746 + 0.258 + 0.395 = 1.399 \text{ Pa}$$

Calculated pressure loss: 1.399 Pa

1-D Heat transfer hand-calculation

Assumption:

- Heat transfer is one-dimensional through the battery thickness.
- Steady-state condition.
- All generated heat is removed by the coolant.
- No heat loss to the surroundings.
- Battery is treated as a homogeneous heat-generating solid.

Calculations:

1. Battery Volume

$$V_b = L W H$$

$$V_b = 0.42 * 0.21 * 0.07 = 0.006174$$

2. Total Heat Generated

$$Q = q' V_b$$

$$Q = 120000 * 0.006174 = 740.88$$

3. Outlet Temperature

$$Q = m c_{(p)} (T_{out} - T_{in})$$

$$T_{out} = T_{in} + \frac{Q}{m c_p}$$

$$T_{out} = 25 + \frac{740.88}{0.012 * 4180} = 39.77$$

4. Coolant temperature rise

$$dT_c = T_{out} - T_{in}$$

$$dT_c = 39.77 - 25 = 14.77$$

5. Average coolant temperature

$$T_{c,avg} = \frac{T_{in} + T_{out}}{2}$$

$$T_{c,avg} = \frac{25 + 39.77}{2} = 32.39$$

6. Cooling interface area

$$A = L W$$

$$A = 0.42 * 0.21 = 0.0882$$

7. Average heat flux

$$q' = \frac{Q}{A}$$

$$q' = \frac{740.88}{0.0882} = 8399$$

8. Maximum Temperature Rise

$$dT_{max} = \frac{q' H^2}{2 k_b} dT_{max} = 120000 * \frac{0.07^2}{2 * 5} = 58.8$$

9. Estimated Maximum Battery Temperature

$$T_{max} = T_c + dT_{max}$$

$$= 32.29 + 58.8$$

$$= 91.19$$

Final result

$$V_b = 0.006174 \text{ m}^3$$

$$Q = 740.88 \text{ W}$$

$$T_{out} = 39.77^\circ\text{C}$$

$$dT_c = 14.77^\circ\text{C}$$

$$q' = \frac{8399 \text{ W}}{\text{m}^2}$$

$$T_{max} = 91.19^\circ\text{C}$$

This means...

- Battery generates about **741 W**
- With **10 g/s** water flow, coolant heats up by about **14.8 °C**
- So water goes from **25 °C** to about **39.8 °C**

CFD Setup

Purpose

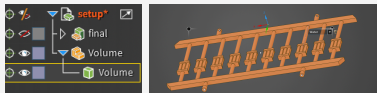
To define the CFD setup, including physical models, boundary conditions, and material properties used for the simulation.

A CAD model created in **Fusion 360** was imported into **SolidWorks** to assemble the battery with the cold plate. The completed assembly was then transferred to Ansys Discovery for simulation. Ansys Discovery 2025 R2 was used to carry out the CFD analysis.

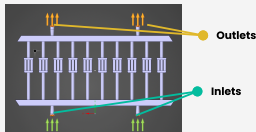
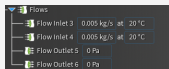
Physical Setup

1. Extract Volume

Volume of the coolant is extracted from the cold plate using volume extract tool.

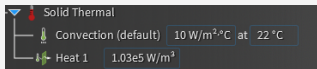


2. Fluid Flow



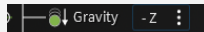
The fluid flow simulation is initialized at **25°C**. The configuration features two mass flow inlets, each set to **0.005 kg/s** at **20°C**, and two pressure outlets maintained at **0 Pa** (static pressure) to define the system's boundary conditions.

3. Solid Thermal



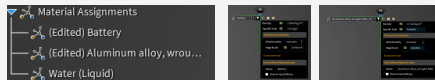
The thermal simulation for the solid components includes a default convective heat transfer coefficient of **10 W/m²·°C** at an ambient temperature of **25°C**. Additionally, a volumetric heat source (Heat 1) is applied at a rate of **1.03e5 W/m³**.

4. Gravity



Gravity was enabled along the **negative Z-axis** to account for **buoyancy** and body force effects, allowing the simulation to better reflect the physical orientation of the system and its influence on fluid density variation and thermal stratification.

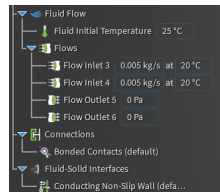
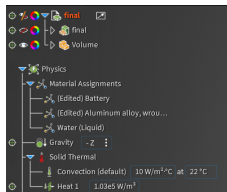
5. Material



Appropriate material properties were assigned to each component:

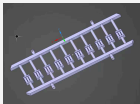
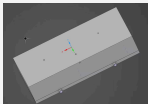
- Battery Module:** Defined as a custom material with a density of **2,530 kg/m³** and a specific heat capacity of **1 kJ/kg·°C**. An isotropic thermal conductivity of **30 W/m-K** was assigned to represent internal heat transfer within the battery.
- Cold Plate:** Assigned the standard material **Aluminum Alloy 6061-T4** with a density of **2,710 kg/m³**. To better capture thermal gradients during operation, both thermal conductivity and specific heat capacity were defined as temperature-dependent properties.
- Coolant:** Assigned the default Liquid Water material properties in the software, with a density of **997 kg/m³** and a thermal conductivity of **0.603 W/m-K**, to simulate heat removal through the cold plate.

Feature Tree



Meshing

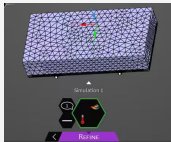
Meshing Approach



The CFD mesh was generated in **ANSYS Discovery** after extracting the internal coolant volume from the **battery module** and cold plate assembly. Figure X shows the overall cold-plate geometry and internal

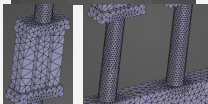
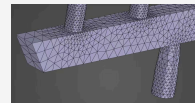
channel layout used in the simulation. The model was then solved in **Refine mode**, where the mesh was generated automatically for the coupled fluid-flow and heat-transfer analysis.

Mesh generation in Refine mode



In this project, the mesh was not manually controlled element by element. Instead, it was generated **automatically** using the built-in meshing process in **ANSYS Discovery Refine mode**, as shown in Figure X. The resulting mesh was then inspected using the mesh display. This approach was suitable for the present study because it provided a **more detailed** solution than Explore mode while maintaining an **efficient workflow** for evaluating the **thermal** and **hydraulic behaviour** of the selected cooling-plate design.

Mesh distribution and critical regions

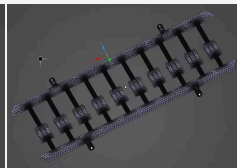
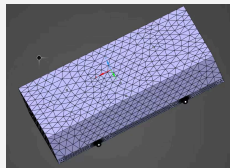


The generated mesh covered both the **coolant flow domain** and the **surrounding solid regions** involved in heat transfer. As shown in the figure, finer mesh resolution was required in **critical regions** such as the **manifold, branch connections, and local geometric transitions**. These locations are important because they are associated with **local flow acceleration, velocity gradients, and pressure losses**. Adequate mesh resolution in these areas was therefore **necessary** to capture the thermal and hydraulic behaviour of the design with reasonable accuracy.

Purpose

To describe the meshing approach, evaluate mesh quality, and assess its suitability for CFD analysis.

Mesh quality summary



The final mesh was inspected in Refine mode using both the built-in mesh display and mesh statistics in ANSYS Discovery, as shown in Figure X. The refined model contained **630,636** elements and **170,989** nodes. Together with the visual inspection of the **overall geometry** and **internal channel layout**, this indicates that the **computational domain** was discretised with sufficient detail to represent the **main flow passages** and coupled **fluid-solid** thermal behaviour of the system. For the scope of this coursework, the mesh density was considered **appropriate** because it was detailed enough to capture the main pressure-drop and temperature-distribution trends while remaining computationally manageable. The final mesh therefore provided an acceptable basis for interpreting the CFD results.

Limitation

1

A full mesh independence study was not carried out, so the solution cannot be described as fully **grid-independent**.

2

The mesh was judged mainly based on **mesh statistics, visual inspection, and the stability** of the refined results, rather than a formal refinement study.

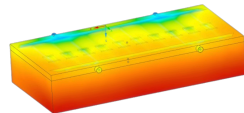
3

The results are therefore **suitable** for design evaluation within this project, but they may not represent the highest possible numerical accuracy.

CFD Output

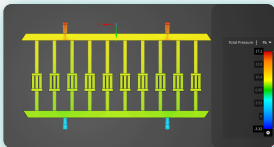
Purpose To present the CFD results, including pressure, temperature distribution, and key performance metrics of the design.

The **pressure** and **temperature plots** illustrate how **fluid flow, heat transfer, and thermal distribution** develop across the design, while the global results **quantify hydraulic resistance, cooling effectiveness, and hotspot behaviour**.



2D Plots

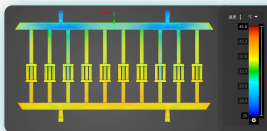
2D Pressure Plots



-3.32 Pa to 17.2 Pa

It shows how pressure is distributed through the plate. **The higher pressure region near the inlet and lower pressure toward outlet** indicate the **pressure drop** as the coolant moves through it.

2D Heat Transfer Plots



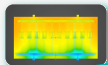
20 °C to 46.6 °C

It shows temperature distribution across the plate. **Cooler regions closer to the flow paths** suggest stronger cooling, while warmer areas highlight where **heat removal is less effective**.

Global Quantities

1

Inlet&Outlet Pressure

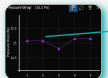


Inlet: 0 Pa

Outlet: 14 Pa

2

Total Pressure Drop

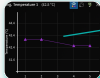


$\Delta P = 15.1 \text{ Pa}$

With minimum of 14.8Pa

3

Average Battery Temperature

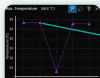


Avg T = 42.5 °C

Peak at 43.1 °C

4

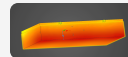
Maximum Battery Temperature



Max T = 46.6 °C

5

Total Heat Removed



Q total = 710.6 W

6

Max Velocity



Max V = 0.146m/s

Comparison & Discussion of Results

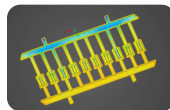
Purpose

To compare the simplified hand calculations with the CFD predictions and evaluate how well the analytical model represents the cooling plate's thermal and hydraulic behaviour.

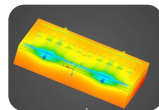
Hand-calculation VS CFD comparison table

	Hand Calculation	CFD Result	Comment
Poorer performance Better performance			
Total Pressure loss	9.8Pa	15.1Pa	Underestimated due to manifold losses
Coolant outlet Temperature	39.77°C	46.6°C	Underpredicted due to non-uniform heating
Coolant temperature rise	14.77°C	26.6°C	Simplified model underestimate heat transfer
Average battery temperature	--	42.5°C	--
Maximum battery temperature	91.19°C	46.6°C	Overestimated from 1D assumption
Heat generate/remove	741W	710.6W	Good agreement with CFD results
Temperature distribution	1-D	2-D	1D model lacks spatial detail

Heat Transfer plots-pipeline



Heat Transfer plots-Battery



Brief Discussion of results

1. Thermal performance

- CFD shows that cooling is **effective**, but not fully uniform across the plate.
- Regions **closer** to the flow path are **cooler**, while others remain **warmer**.
- This means **temperature uniformity** should be considered in addition to maximum temperature.
- The 1-D model is **useful** for quick estimation, but it cannot identify **local hot spots**.

2. Hydraulic performance

- The **pressure distribution** from CFD suggests that most of the pressure is **lost** along the **flow passages** and around the **manifold transitions**, rather than being evenly consumed everywhere.

3. Trade-off

- Better cooling** often requires stronger flow interaction, which can increase **pressure loss**.
- The aim is to reach an **optimal compromise**, not simply the minimum of one parameter.
- The current design appears to provide a **reasonable balance** between thermal and hydraulic requirements

4. Overall evaluation

- Further work should focus on **optimization**, especially **temperature uniformity** and local flow losses.

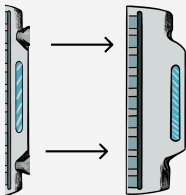
Future Recommendation

Purpose

To propose future design improvements and summarise the overall performance of the cooling plate concept.

Future Improvements

1. Increase Manifold Area



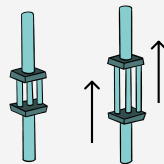
The design can be improved by **widening** the manifold region to reduce **hydraulic resistance**. Narrow inlet and outlet headers increase **coolant velocity** and create **higher local losses**, which raises **pressure drop**. A larger manifold allows smoother **flow distribution** into the channels, helping to lower pressure drop while maintaining good cooling performance.

3. Add more tapering



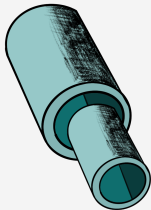
The design can also be improved by reducing **sudden contractions, expansions, and sharp turns** within the flow path. These geometric changes create additional **minor losses**, which increase the **overall pressure drop**. Introducing **tapered transitions** and **smoother corners** allows the coolant to move more **gradually** through the cold plate, reducing **local resistance** and improving **flow stability**. This refinement is effective because it can lower pressure drop without significantly affecting T_{max} .

4. Increase the length of micro channel



Increasing the **length** of the microchannels may help reduce **T_{max}** by extending the heat transfer path and increasing coolant contact. However, longer channels also increase frictional losses, which can raise the overall pressure drop.

2. Increase Channel Width – T_{max} Trade off



The **pressure drop** can also be reduced by slightly increasing the **channel width**. Very narrow microchannels significantly increase **frictional losses**, leading to **higher resistance**. By widening the channels, the flow resistance can be lowered. This approach trades a small amount of **thermal performance** for a more efficient and manageable pressure drop.

Conclusion

1

Balanced Overall Performance

The final cold plate shows a **good compromise** between cooling effectiveness and hydraulic efficiency, meaning it removes heat well without high flow resistance

2

Effective hotspot cooling

A **T_{max} of 42.5°C** suggests that the design is able to control **peak temperatures** especially in the high-heat region

3

Moderate Pressure Drop

Pressure drop of **15.1 pa** indicates that the coolant experiences **resistance** as flowing through, but the value is still within a moderate range

4

Further optimisation is possible

Although the current performance is **strong**, there is still room to improve the **hydraulic side** of the design by reducing unnecessary **flow resistance**

Team & Project Management

Team Contribution

- Yuxuan Zhou – Modelling and CAD development
- Cici Song – Project planning and concept development
- Joe Chen – Hand calculations and related analysis
- Simon Xia – Sketching and visual concept communication

Slide Contribution	Cici	Yuxuan	Joe	Simon
SUV Car Design				
Project Summary	●	●		
Printed Model		●		
Wind-tunnel Testing	●			
Result Comparison			●	
Design Optimisation				●
Pipeline Design				
Design Summary		●		
Bechnmarking	●			
Modeling		●		
Pressure loss Calculation			●	
Heat Transfer Calculation			●	
CFD Set-up	●			
Meshing	●	●		
CFD Output				●
Calculation-CFD Compare			●	
Discussion of Result			●	
Future recommendations		●		●
Conclusion				●
Project Management	●			
References				●

Project Planning

SUV Car

Action

Slides

Week 8	Week 9	Week 10	Week 11
Conduct Wind Tunnel Test	Analyze Results	Optimisation	Conclusion
• Do wind-tunnel test. • Model was selected and printed in week 7. • Wind-tunnel test results received. • Analyze result and recalculate C_d from area.		• Analyze the result. • Compare with CFD simulation • Optimise Design	• Validate optimised design • Do the conclusion
Project Summary; Printed Model	Wind-tunnel Testing	Result Comparison; Design Optimisation	Validation; Cover Page

Cold Plate Pipeline

Action

Slides

Week 8	Week 9	Week 10	Week 11
Concept Finalisation	Modeling & Calculation	Conduct CFD	Analysis & Conclusion
• Bechmarking • Improve selected design • Sketch the new design • Give sizes and dimensions.	• Model the design using fushion 360 • Confirm Specific dimensions • Do pressure loss and heat transfer calculation	• Do the CFD simulation • Fix any problem encountered in CFD • Analyze the result • Compare results	• CFD learnings • Future improvements • Project management
Benchmarking	Modeling; Pressure loss Calculation; Heat transfer Calculation	CFD Setup; Meshing; CFD Output; Calculation-CFD Compare	Result Discussion; Future recommendation; Conclusion; Project Management

Reflection

Overall, the project showed strong teamwork and effective group management, as clear responsibilities, consistent communication, and structured planning helped the team collaborate efficiently and respond well to challenges.

Reference

<https://www.racecar-engineering.com/cars/ferrari-sf90/>

<https://www.press.bmwgroup.com/global/article/detail/T0276225EN/the-new-bmw-i8-roadster-the-new-bmw-i8-coupe?language=en>

https://docs.google.com/spreadsheets/d/1EJLm_p-kKseYEs_aj2laomDhptKJxnJ/edit?usp=drive_link&ouid=118051373221590008288&rtpof=true&sd=true

CFD Graphs

