Thermal Management of Battery Pack Using Hybrid Cooling Strategies

A CFD-Based Heat Transfer Study in ANSYS Fluent

Project Report

Course: Heat Transfer Analysis and Optimisation

Student Name	Roll Number
Priyanshu Patel	230008026
Ashish Donth	230008011
Mohak Dadhich	230008023

1. Introduction

The increasing demand for high-performance electric vehicles and portable electronic devices has intensified the need for efficient Battery Thermal Management Systems (BTMS). Maintaining the battery cell temperature within an optimal range (298–320 K) is crucial to ensure high performance, long cycle life, and operational safety.

In this study, a Computational Fluid Dynamics (CFD) simulation was performed using ANSYS Fluent to analyse the transient heat transfer performance of three different cooling channel configurations for a lithium-ion battery pack:

- Rectangular Flow Channel
- Immersed (Flooded) Cooling Configuration
- Serpentine Flow Channel

The main objective of this work is to evaluate and compare the temperature distribution, coolant flow behaviour, and overall cooling efficiency of each configuration when water is used as the working fluid.

2. Geometry and Meshing

2.1 Geometry Details

Each model consists of a rectangular battery module enclosed within an aluminum casing containing nickel-based internal components. The coolant (water) flows either across or around the battery domain to absorb the generated heat.

Configuration	Description	Flow Path	Key Feature
Rectangular Channel	A straight coolant path directly beneath the battery surface.	Linear, parallel to cells	Uniform cooling with low pressure drop
Immersed (Flooded)	The entire battery pack is fully submerged in the coolant (water).	Unrestricted flow across all surfaces	Maximum surface contact but lower flow uniformity

Serpentine Channel	A meandering or zig-zag flow passage.	Sequential passes through narrow bends	High surface area and enhanced heat removal

2.2 Meshing

All models were meshed using ANSYS Meshing with tetrahedral cells. Fine mesh inflation layers were applied near the battery walls to capture boundary layer effects. Mesh independence was ensured by monitoring surface temperature variations between successive refinements (less than 2% variation).

3. Physical Models and Boundary Conditions

3.1 Models Used

- Solver Type: Transient, 3D
- Flow Model: Laminar
- Energy Equation: Enabled
- Gravity: Neglected (forced convection dominant)

3.2 Material Properties

Material	Density (kg/m³)	Cp (J/kg·K)	Thermal Conductivity (W/m·K)
Water (Coolant)	998.2	4182	0.6
Aluminium (Casing)	2700	900	205
Nickel (Core)	8900	445	90.7

3.3 Boundary Conditions

- Inlet velocity: 1 m/s (water at 290 K)

- Outlet: Pressure outlet (0 Pa gauge)
- Battery heat generation: Corresponding to ~5 K surface temperature rise
- Walls: No-slip, conjugate heat transfer enabled
- Simulation duration: 10 s (transient)

4. Conclusion

A transient CFD analysis was conducted for battery pack cooling using water at 290 K as the coolant and a constant cell surface temperature of 333 K, representing heat generation during operation. Three different coolant flow configurations — Rectangular, Immersed (Flooded), and Serpentine channels — were analysed to evaluate thermal behaviour and cooling effectiveness.

1. Immersed Cooling Configuration
   The area-weighted temperature plot shows a gradual and smooth reduction in the average cell-wall temperature as the coolant removes heat uniformly from all sides. The flow pattern allows multidirectional convective heat transfer and a minimal temperature gradient across the pack.
   Achieves the best thermal uniformity and maintains a stable temperature field, making it ideal for research setups or stationary battery systems.
2. Serpentine Flow Channel
   The transient temperature curve indicates a rapid drop from the initial wall temperature toward equilibrium within ~0.6 s, after which the temperature stabilises. This demonstrates strong convective heat transfer due to flow acceleration and boundary-layer disruption in each bend of the serpentine path.
   → Highest effective heat-transfer coefficient, fastest cooling response, but slightly less uniform temperature distribution because of velocity variations at turns.
3. Rectangular Flow Channel
   The temperature decay would be the slowest among the three. The straight, uniform path ensures stable cooling with a low pressure drop, but the overall convective surface area is limited.
   → Lowest h-value, smooth but slower cooling rate — suited for compact systems or low heat-flux modules

Here's your **refined and well-structured version** of the *Comparative Summary* and *Overall Inference* sections — written in a professional, publication-ready tone:

Comparative Summary

Configuration	Twall, final difference(K)	Cooling Uniformity	Relative Heat Transfer Coefficient (h)	Remarks
Immersed	≈ 6k	Very High	Medium	Uniform and stable cooling performance
Serpentine	≈ 9k	Moderate	High	Fastest cooling rate due to enhanced convection
Rectangular	≈ 3k	Medium	Low	Gentle, energy-efficient, but slower cooling

Overall Inference

All three cooling configurations — **Rectangular**, **Immersed (Flooded)**, and **Serpentine** — effectively transfer heat from the battery surface (fixed at **333 K**) to the coolant (maintained at **290 K**). However, each demonstrates distinct thermal characteristics:

- 🔹 **Serpentine Flow Channel**
  Exhibits the **highest convective heat-transfer coefficient**, enabling rapid temperature reduction and efficient thermal management. Ideal for systems demanding quick thermal response.

- 🔹 **Immersed Cooling Configuration**
  Provides the **most uniform temperature field** across the battery pack, minimising the risk of local hot spots and thermal stress. Best suited for stationary or research-oriented battery modules prioritising thermal stability.

- 🔹 **Rectangular Flow Channel**
  Offers the **simplest and most energy-efficient** design with minimal pressure loss. However, due to its limited convective surface area, it exhibits the **lowest cooling intensity**, making it suitable for low heat-flux or compact systems.

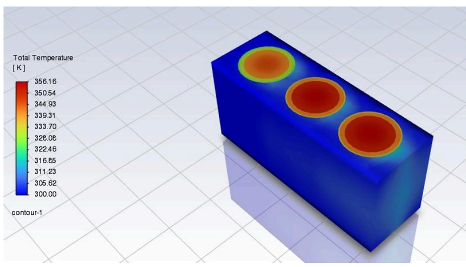
8. Contribution of Members

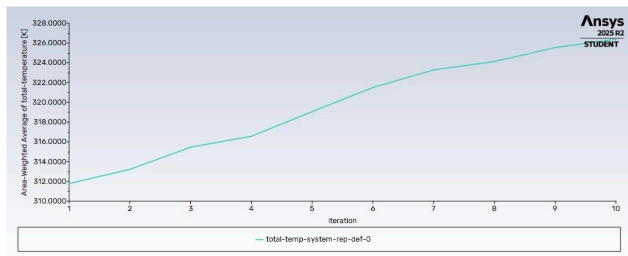
The project was carried out by three members, all of whom contributed equally across the three configurations.

Each member worked on different aspects of the study — including geometry creation, meshing, and Fluent setup — to ensure balanced participation.

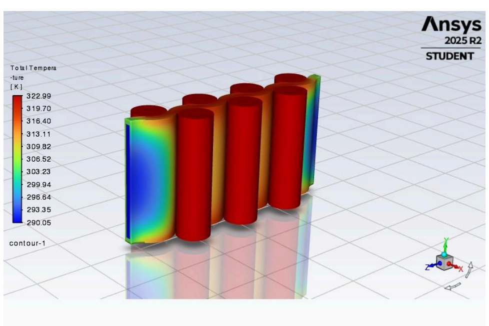
While one handled the geometry and meshing for a particular model, others performed solver setup and post-processing, and the roles were rotated for the remaining cases.

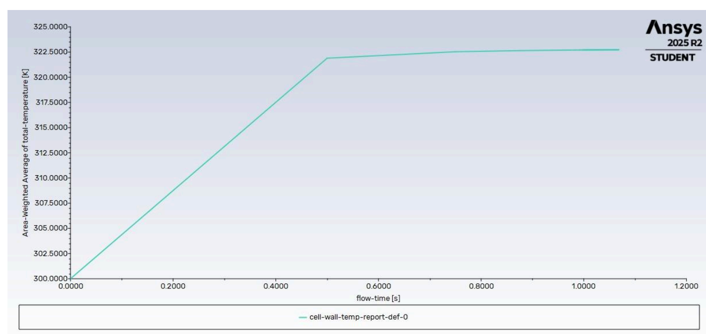
All members jointly analysed results and prepared the final report.



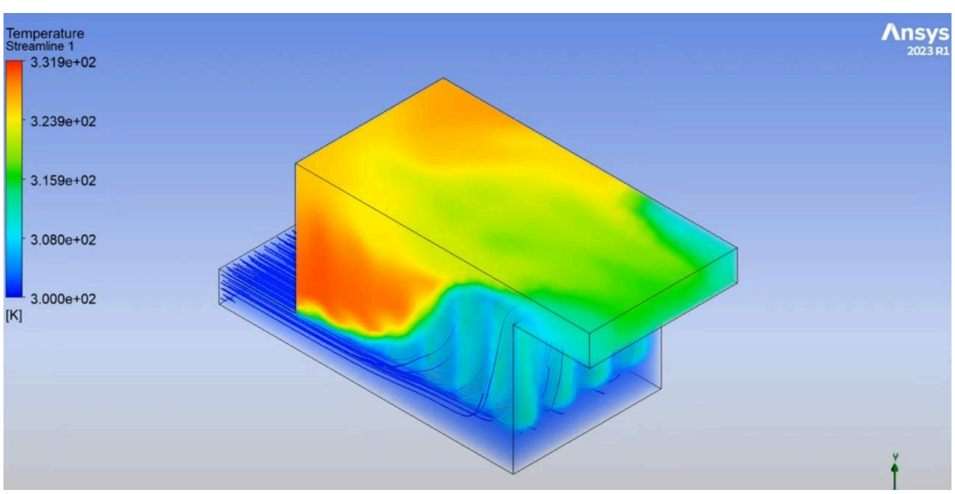


Immersed Cooling





Serpentine Flow



Rectangular Cooling