2D CFD Analysis for Optimizing Air Cooling in EV Battery Packs

Ancy Sijo Kureekal
Gustaf Larsson
Tanuj Nagentra Ramachandran

Lunds Tekniska Högskola
MMVN05 - Numerical Fluid Dynamics and Heat Transfer
Report

The authors certify that all material in this report that is not the author's own, is clearly identified and that no part of the material is previously published.

2D CFD Analysis for Optimizing Air Cooling in EV Battery Packs

June 8, 2025

Abstract

The increasing trend of transition to electric vehicles(EVs) makes it important to study the battery thermal management system(BTMS), as lithium-ion batteries are highly sensitive to variation in temperatures. To enhance the BTMS performance in EVs , this study investigates the effect of the cell arrangement and inlet airflow on heat dissipation in the battery module. A 2D CFD model of 18650 lithium-ion battery was developed in ANSYS Fluent, where two configurations: aligned and staggered, were simulated under varying inlet velocity. A structured mesh of element size 0.001m was used, and the study also conducted the mesh sensitivity to confirm the adequacy of the mesh. Results showed that increasing the inlet velocity led to the reduction in cell temperatures in both configurations. Among them, the Staggered configurations offered better thermal uniformity and minimized hotspots. This highlights that airflow velocity and cell arrangement are key factors in designing efficient air-cooled battery systems.

Keywords: Battery Thermal Management System(BTMS), Computational Fluid Dynamics (CFD),2D Simulation, Air Cooling, EV Battery Cooling

Contents

1 Introduction

The demand for electric vehicles (EVs) is currently increasing since it is gaining attention as a sustainable alternative to conventional transportation. This has therefore intensified the need for research in battery thermal management systems (BTMS). Lithium ion batteries are widely used in EVs due to their advantages such as high energy density, long cycle life and high efficiency. However, these batteries are very sensitive to temperature variations. Improper heat dissipation could result in uneven temperature distributions, reduced efficiency and also could lead to thermal runaway which is a hazardous condition where the excessive heat triggers a chain reaction generating more heat and potentially resulting in fire or explosion.

There are various thermal management strategies such as air cooling, liquid cooling and phase change materials (PCM) among which air cooling remains the most widely used technique. It is cost effective, more simple and reliable. However, a significant challenge with this method is maintaining uniform cell temperatures under high discharge rates and compact packaging conditions.

A study by Wang et al. (2017) conducted the 3D simulation of a battery pack using ANSYS FLUENT. They experimented with parameters such as cell spacing and inlet velocity for different cell arrangements and analysed the cooling performance. The results showed that a staggered cell arrangement offered better thermal uniformity compared to the aligned configurations under higher airflow conditions. Similarly, Bamrah et al. (2022) performed a CFD analysis using 3x8 configuration and various flow velocities. Their study also concluded that staggered cell arrangement with increased inlet velocity improved the temperature uniformity and reduced hotspots across the pack. Even though 3D modeling exists in previous researches, a simplified 2D modeling could offer faster computation and theoretical validation for comparative studies. In this report, a 2D CFD analysis is done in order to study the influence of cell arrangement and varying inlet velocities on the thermal management of a battery pack that uses an air cooling technique. This study uses a validated 3D simulation by Wang et al. (2017) as a reference for verification since there was very limited availability of 2D references.

The aim of this study is to understand the airflow behavior and its thermal impacts in battery modules and also to provide an insight into how the geometrical modifications and boundary conditions impact heat dissipation and thermal uniformity using the simplified 2D model.

2 Methodology

The methodology, as shown in figure 1, followed a step-by-step CFD approach using ANSYS Fluent to analyse the thermal performance of an EV battery pack. The process began by designing the battery pack geometry for different cell arrangements. The governing equations were defined with relevant assumptions and the boundary conditions were set. A mesh was then generated and was used to run the simulations for various airflow conditions. After which, the results were analysed and modifications were made if the results were not optimized. This cycle was repeated until a satisfactory performance was achieved.

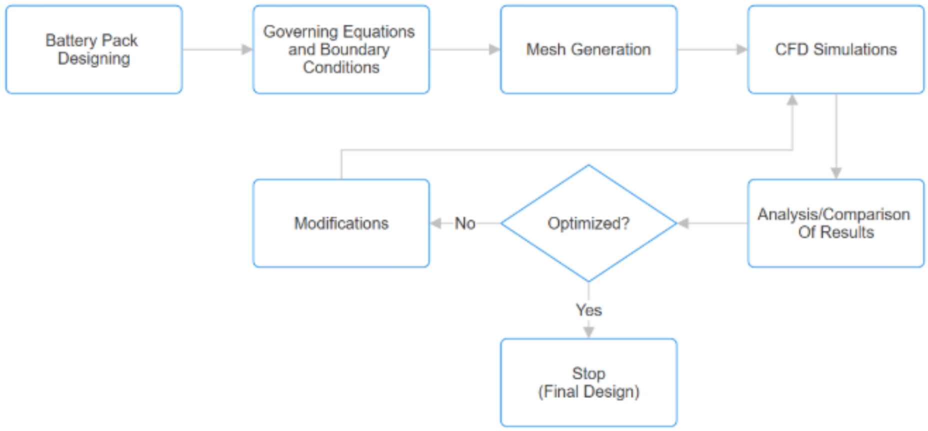


Figure 1: Flowchart for the CFD analysis

3 Theory

Heat production models for batteries are rather complicated. The thermal energy generated and absorbed by a battery is a function of charge and discharge rates, chemical composition of the battery, ohmic resistance and potential, just to mention a few variables. But since we are only interested in the cooling effect of air as a function of velocity, we have made the simplification that there is a constant heat flux from the surface of the battery cells and no thermal radiation.All our assumptions are listed below:

Assumptions:

- Ideal gas.
- Steady state
- Constant heat flux from batteries.
- No thermal radiation
- Newtonian fluid (constant viscosity)
- No external forces

How these assumptions affect the governing equations are derived below. The only heat transfer involved in our study is heat conduction and can be expressed as Fourier's law (Versteeg et al. 2007) :

$$q_{\text{conduction}} = -k\nabla T \tag{1}$$

where $q_{\text{conduction}}$ is the heat flux vector, k is the thermal conductivity, and ∇T is the temperature gradient.

Assuming incompressible gas the continuity equation becomes:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \Rightarrow \nabla \cdot u = 0 \tag{2}$$

Newton's second law applied to a fluid:

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot \rho u u = -\nabla p + \nabla \cdot \boldsymbol{\tau}_{ij} + \rho f \tag{3}$$

For a incompressible fluid with constant viscosity the equation becomes:

$$\frac{\partial u}{\partial t} + \nabla \cdot u u = \frac{-\nabla p}{\rho} + v\nabla^2 \cdot u \tag{4}$$

The energy equation expressed with regards to the temperature is:

$$\frac{\partial(\rho C_p T)}{\partial t} + \nabla \cdot (\rho u C_p T) = Q_r - \nabla \cdot q + \nabla \cdot (\boldsymbol{\tau} \cdot u) + \rho f u + \frac{\partial p}{\partial t} \tag{5}$$

With our assumptions of incompressible fluid and no thermal radiation and insert of equation 1 the energy equation can be simplified to:

$$u \cdot \nabla T = \alpha \nabla^2 T \tag{6}$$

where $\alpha = \frac{k}{\rho C_p}$

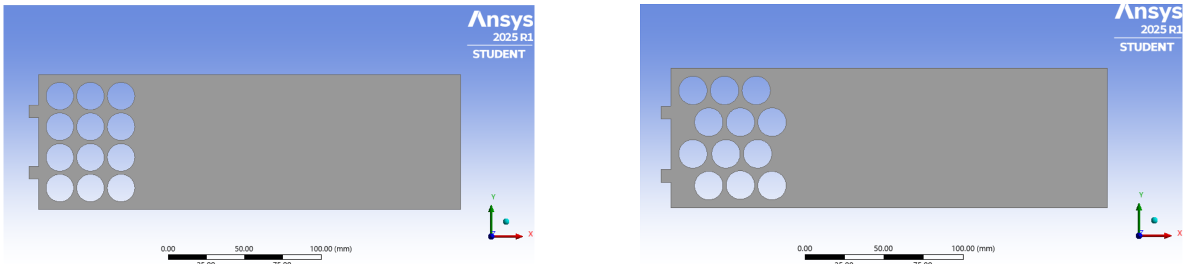
For the simulation we used the SIMPLE scheme and how the governing equations where discretized are summarised Table 1 below. The pressure was calculated using a second order central scheme and the energy and momentum equation was expressed using second order upwind. Second order schemes reduce the numerical diffusion, especially where the flow is not aligned with the grid lines but could cause problems with convergence (Versteeg et al. 2007). The transient was set to first order Implicit, which means that it will be unconditionally stable even for large timesteps.

Table 1: Discretization scheme

Governing equations	
Pressure	Second order
Momentum	Second order upwind
Energy	Second order upwind
Transient Formulation	First order implicit

4 Set-up

The simulation was carried out for the 2D model of lithium ion battery pack with 18650 type cylindrical cells. The domain consisted of 12 cells in total each with a diameter of 18 mm and the total domain had a horizontal length of 276 mm and a vertical length of 88 mm. As shown in figure 2a and 2b, the cells were arranged in both aligned and staggered configurations with an inter-cell spacing of 2 mm between them to allow the airflow between the cells. Two 8



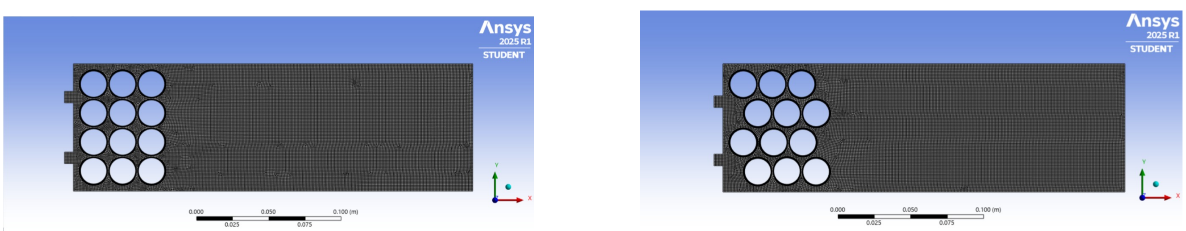
(a) Aligned Geometry

(b) Staggered Geometry

Figure 2: Aligned (a) and staggered (b) geometries for battery thermal management analysis.

mm wide inlets were introduced on the left side of the domain and were separated by a 32 mm gap to simulate the forced convection cooling.

The 2D domain was discretized using a structured mesh as shown in figure 3 with finer grid near battery surfaces and inlet region to get more accurate temperature and flow gradients. A global element size of 0.001m was used for the creation of the mesh. Additionally, Edge sizing with the same element size was added around the edges of the battery cells for effective local resolution with a growth rate of 1.1. Similarly, inflation layers were added with an initial thickness of 0.001m to capture near-wall flow and thermal gradients.



(a) Aligned Configuration Mesh

(b) Staggered Configuration Mesh

Figure 3: Aligned (a) and staggered (b) mesh.

A pressure-based solver under steady state condition was used for the CFD simulation. Effects due to gravity were neglected since the airflow in the horizontal direction dominates and so gravity was set to zero. The energy equation was enabled to consider the heat transfer and the turbulence was modeled using the standard k-ε model with enhanced wall treatment and curvature correction.

Material properties for lithium ion cells were referenced from Wang et al. (2017) with a density of 7187 kg/m³, specific heat capacity taken as 1210 J/kg·K and the thermal conductivity as 2.853 W/m·K. Air was used as the working fluid and default properties were taken from the ANSYS database.

For boundary conditions, the inlet temperatures were kept at 293 K and inlet velocities were set as 1 m/s which was later varied to 3 m/s, 5 m/s, 7 m/s and 9 m/s to analyse their impact on cooling. A gauge pressure of 0 Pa was defined at the outlet. A uniform heat flux of 32.35 W/m² was applied to the cell surfaces to represent the heat generated during the battery discharge (this value is derived for the 2D model from the heat generation rate 7162 W/m³ corresponding to a 3C discharge rate as mentioned in the reference paper by Wang et al. (2017)).

5 Mesh Sensitivity Analysis

A mesh sensitivity analysis was performed for the staggered configuration by varying the element size and thereby measuring the average temperature of the first 6 cells. Simulations were carried out using three different element sizes: 0.003 m, 0.002 m, and 0.001 m, keeping all the other parameters constant. As shown in figure 4, variation of average temperature across different element sizes was minimal, especially between 0.002 m and 0.001 m. This indicates the fact that further mesh refinement will not produce any significant difference in the outcome. Therefore, an element size of 0.001 m was chosen as it gives a good balance between computational cost and accuracy.

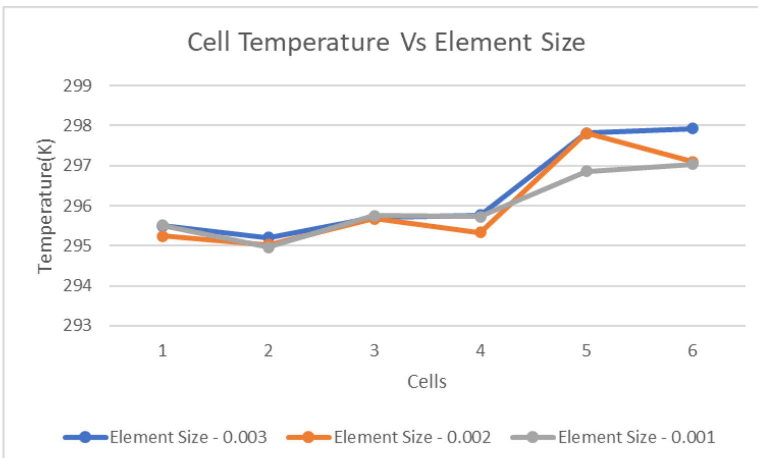


Figure 4: Grid sensitivity analysis for staggered cell arrangement

6 Turbulence Modeling

For the CFD simulations, the standard k-ε turbulence model was chosen initially as this was also used in the reference study. This method provided advantages such as low computational cost, better convergence and reliable performance in our case. However, to further evaluate the results, we also tested the set up with SST k-ω. Although the SST k-ω is known for improved accuracy, in our case this method did not fully converge. A probable reason for the solution to not converge might be the lack of a finer mesh near the walls. As a result, the reliability of the results with this method is questionable even though a similar temperature trend was obtained as in the k-ε model as shown in figure 5. Therefore we decided to finalize and use the k-ε turbulence model for our simulations.

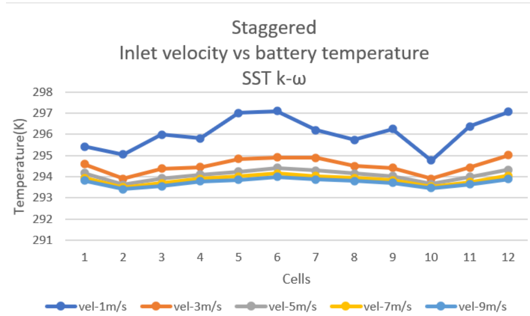


Figure 5: Turbulence modeling for staggered cell arrangement

7 Results and Discussions

The battery module of two different geometric configurations, staggered and aligned, was simulated under varying inlet air velocities from 1 m/s to 9 m/s. The static temperature for each cell was measured using the surface integral method, and the graph for each case was plotted. Additionally, contour plots for each case were also generated to visualize the temperature gradients and the flow behaviour across the module.

7.1 Aligned Cell Arrangement

In this configuration, as shown in fig 12, the cooling performance of the module improved with an increase in the inlet velocity. When the inlet velocity was 1 m/s, the battery pack had higher cell temperatures mostly around the central cells. This is further confirmed by the temperature contour plot shown in fig 7a, where red and orange zones (higher temperature zones) are more near the battery surfaces. This indicates the inefficient heat dissipation and poor thermal uniformity.

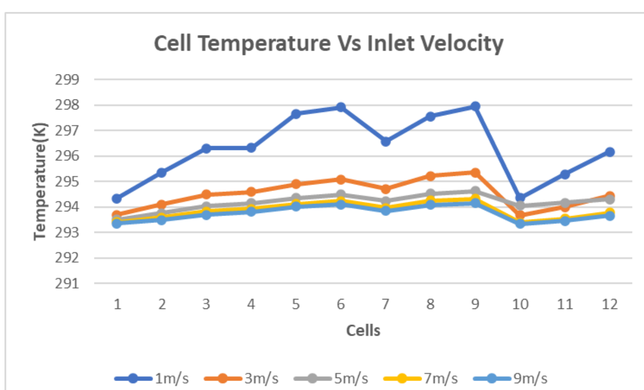
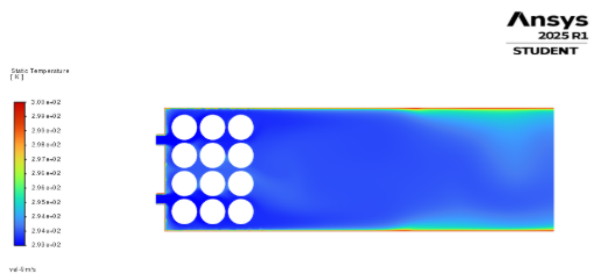
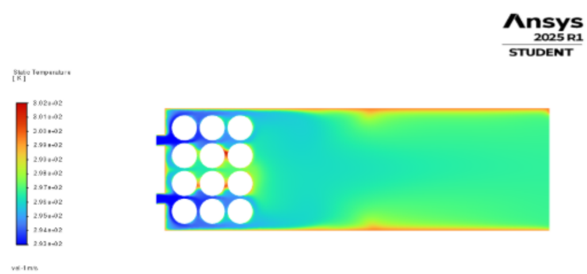


Figure 6: Variation of cell temperature with different inlet air velocities (Aligned).

As the airflow increases the cooling performance also increases which in turn reduces the surface temperatures. It can also be noted that there is not much change in cell temperature when the velocity is increased beyond 5 m/s. At 9 m/s, as shown in the contour plot Fig 7b, the zone around the battery surfaces are mostly blue indicating that the heat is effectively removed. It can be seen that minor gradients are present near the center and this suggests uneven airflow distribution even at higher inlet velocities.



(a) Inlet velocity of 1 m/s



(b) Inlet velocity of 9 m/s

Figure 7: Temperature Contour Plots (Aligned).

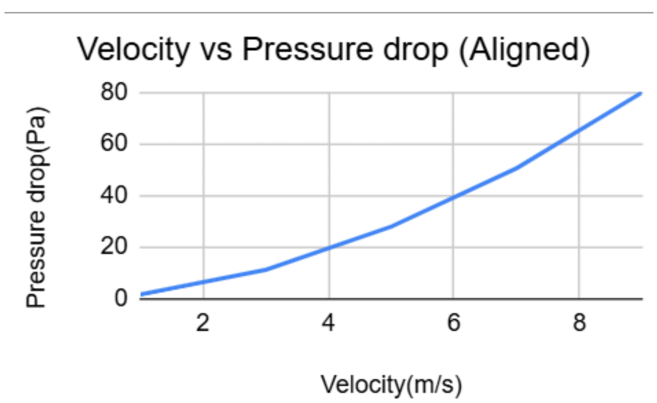


Figure 8: Pressure drop (Aligned).

The pressure drop in the aligned configuration showed a consistent increase as the inlet velocity increased, as shown in figure 8. At the highest velocity of 9m/s, the pressure drop reached approximately 75 Pa. This indicates the higher flow resistance in this arrangement. Therefore it can be concluded that the aligned arrangement demands more energy to maintain higher velocities due to increased obstruction.

7.2 Staggered Cell Arrangement

In the staggered configuration, the turbulence increases due to the airflow path and this, in turn results in better mixing. Due to which a more uniform thermal distribution can be seen even at lower inlet velocities. At 1 m/s, the surface temperatures were slightly lower and were more evenly distributed when compared to that of the aligned configuration as in fig 10a.

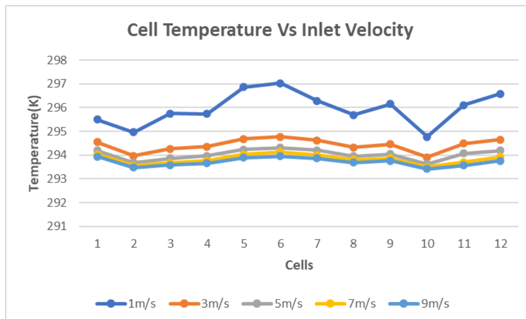
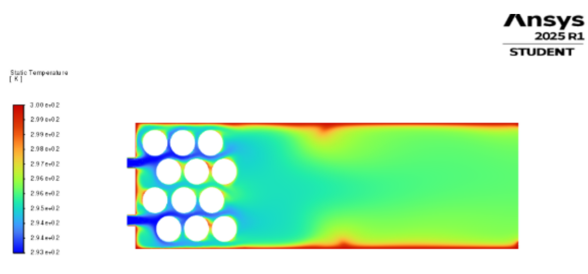
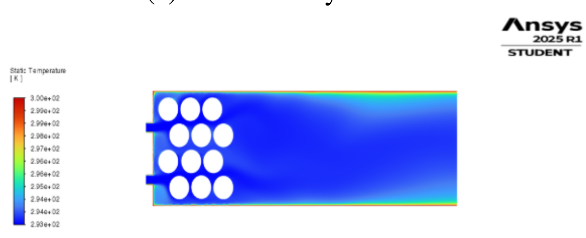


Figure 9: Variation of cell temperature with different inlet air velocities (Staggered).



(a) Inlet velocity of 1 m/s



(b) Inlet velocity of 9 m/s

Figure 10: Temperature Contour Plots (Staggered).

As the inlet velocities increased, the staggered arrangement had better results in terms of peak temperatures as shown in fig 9. The contour plot at 9 m/s is mostly blue near the cells and has minimal thermal gradients throughout the domain as in fig 10b. This suggests better cooling and suppression of localized hotspots especially in the region where the airflow was restricted in the aligned configuration.

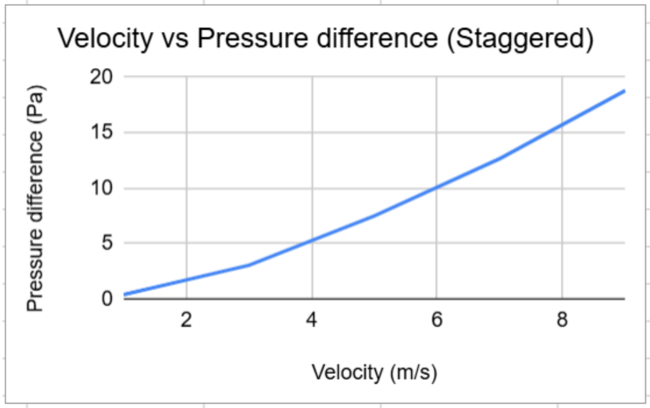


Figure 11: Pressure drop (Staggered).

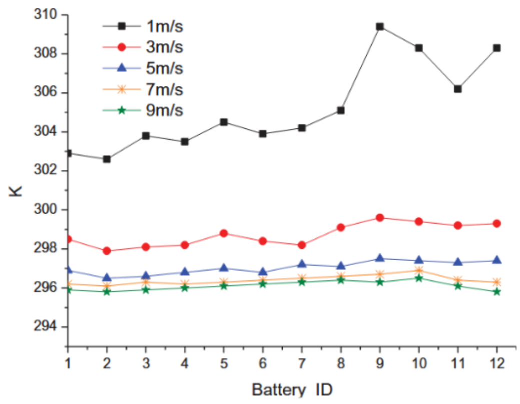
The pressure drop in the staggered configuration showcased a much lower pressure drop across the various velocities which were tested. The highest pressure drop in this cell arrangement was at 9 m/s with a value of 18 Pa as shown in figure 11 which is much lesser when compared to the aligned configuration. This was attained due to lesser flow resistance in the setup thereby providing better airflow paths making it more energy efficient while providing effective cooling.

8 Validation

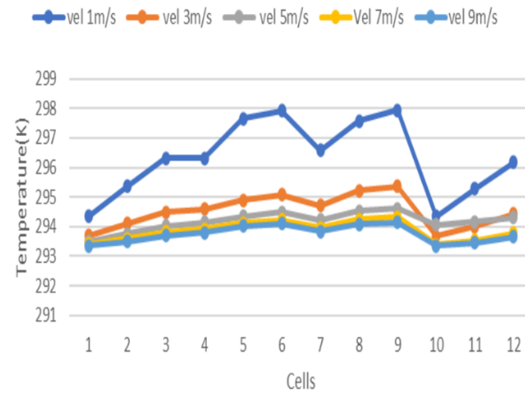
This study conducted a qualitative validation, as illustrated in figure 12, based on the work of Wang et al. 2017.

Looking at the details, it is evident that the result differs between the two studies. The differences can be due to many things. For example, the study conducted by Wang et al. 2017 simulated a battery pack in three dimensions and included a thermal model for the battery that took ohmic and polarization resistance into account. In contrast the present study modeled the battery pack in two dimensions and simplified the thermal model of the battery by assuming constant heat flux. It is challenging to understand what will happen to three-dimensional quantities, such as density, when you dissect the problem into a two dimensional framework.

But if we take a broader look at the result and overlook the details we can see that the inlet velocity has a diminishing return on the cooling of the battery cells.



(a) Variation of cell temp from the reference study *(Wang et al. 2017)*



(b) Variation of cell temp from this study

Figure 12: Qualitative Comparison of the cell temperatures

Our study aligned with the conclusion of Wang, Fan and Liu where they concluded that the inlet velocity less than 5 m/s "can promote the cooling performance of battery remarkably" and that at inlet speed above 5 m/s, "the promotion of the cooling performance is not obvious".

9 Conclusion

This study conducted a 2D CFD analysis for optimizing air cooling in EV battery packs by comparing both aligned and staggered configurations. The results conclude that increasing the inlet velocities improved the cooling performance as a result of the convective heat transfer. Among the two cell arrangements, staggered configurations performed better than the aligned configurations in terms of thermal uniformity and peak temperature reduction. Additionally, the pressure drop was observed to be significantly lower in the staggered when compared to the aligned which also resulted in improved energy efficiency. It was also observed that increasing the inlet velocity beyond 5 m/s resulted in minimal changes in cell temperature, therefore increasing the velocity is not required as this can lead to higher pressure drop. These findings gives insight into the importance of optimizing both the cell arrangement and airflow conditions in order to achieve an efficient and reliable thermal management in battery systems.

References

Bamrah, Parush et al. (2022). “CFD analysis of battery thermal management system”. In: *Journal of Physics: Conference Series* 2178.1, p. 012035. DOI: 10.1088/1742-6596/2178/1/012035.

Versteeg, H.K. et al. (2007). *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. 2nd. Chapter 5, p. 150. Harlow, England: Pearson Education.

Wang, Zhenpo et al. (2017). “Simulation of temperature field of lithium battery pack based on computational fluid dynamics”. In: *Energy Procedia* 105, pp. 3339–3344. DOI: 10.1016/j.egypro.2017.03.764.