Numerical Simulation of Heat Transfer Characteristics of Aluminium Heat Sink for Electric Vehicle Battery Cooling Using ANSYS Fluent

Sakshi R. Mohod, Prajwal R. Chute, Bhojraj S. Rathod, Prajay C. Gharde, Vedant S. Ambulkar
Mechanical Department, Priyadarshini College of Engineering, Nagpur, Maharashtra, India.
Corresponding Author: **Dr.K.R Aglawe**

Abstract

Thermal management plays an important role when designing batteries for Electric Vehicles (EV), to ensure they can operate efficiently, safely and for a long time. This paper looks at a simplistic cooling system using an Aluminium Heat Sink with an air duct. The heat transfer characteristics have been looked at using ANSYS Fluent, a CFD software. Three cases of air inlet velocities have been analysed; 1m/s, 2m/s and 3m/s. The highest and lowest temperatures found were 309K, 306K, 305K, 302K, 302K and 299K respectively. This paper shows temperatures in Kelvin. However, other parts of the report mention temperatures in Celsius. This needs to be noted as the temperatures can be found in the files from Fluent. Overall, the higher the velocity the lower the temperatures. This paper shows how CFD can be used to analyse the cooling conditions. However, several recommendations can be made. This includes validating the design, ensuring mesh independence and that all heat inputs and boundary conditions are fully documented.

Keywords: Electric Vehicle, Lithium Ion Battery, Aluminium Heat Sink, Air Cooling, Computational Fluid Dynamics, ANSYS Fluent, Thermal Management.

1. Introduction

Electric vehicles are becoming more of an alternative to vehicles using an internal combustion engine. Being able to keep the battery pack of the vehicle at the correct temperature range is key. During charging and discharging heat can be generated due to internal resistance and electrochemical reactions. If heat isn't removed efficiently from the battery this can cause temperature gradients and therefore faster deterioration or safety issues. Therefore thermal management is key.

There are many ways that the battery can be cooled. This can include forced air, liquid, phase change and hybrid cooling. Air cooling can be a good method to start with due to its simplicity and there being no liquid circuits. However, the performance will depend on the air distribution, duct design, inlet conditions, heat sink design and heat load.

Using a heat sink improves the heat transfer by providing a conductive path for the heat to travel along and increasing the surface area. Aluminium is commonly used due to it having good thermal conductivity and low density. The duct ensures the air flows over the heat sink surfaces and some of the heat will be removed. The aluminium heat sink was analysed using ANSYS Fluent to see how the temperatures would behave with three different inlet air velocities.

2. Aim & Objectives

To numerically analyse the heat transfer characteristics of an aluminium heat sink including an air duct for potential battery cooling in an electric vehicle.

Create a 3D model of the air duct and aluminium heat sink. Identify regions of solid and fluid and create mesh.

Assign correct materials, flow conditions and thermal conditions in ANSYS Fluent. Analyse the temperature distribution for the three different inlet air velocities.

Compare the temperatures and identify the case with the lowest maximum temperature.

3. Literature Review

Many studies have been done on cooling systems for batteries. Air cooling can be a good option as it is simple to implement and have low complexity. Liquid cooling however can provide better cooling. The best option will depend on the type of battery, duty cycle, packaging, cost and acceptable temperature variation.

The heat sink material and design will impact the performance. The fin height, thickness, spacing and orientation will affect the surface area and resistance to airflow. Increasing the air velocity can improve the convective heat transfer however can cause high pressure drops and increase fan power requirements. The design should be balanced between reducing the temperature and the amount of energy required to move the air.

CFD can help engineers understand how the airflow will interact with the heat transfer before creating a prototype. The flow and energy equations are solved inside the discretised domain using ANSYS Fluent. The accuracy of the results will depend on the geometry, mesh, material data, boundary conditions, physical models and ensuring the solution has converged. See references at the end of this report for some background information on battery cooling and numerical thermal analysis. Please check the full publication details.

4. Methodology

4.1 Geometry & Domain

The aluminium heat sink was placed inside an air duct to ensure the air flows through. The aluminium will conduct the heat away. There will be a solid region for the heat sink and a fluid region for the air. The exact fin sizing should be mentioned but this wasn't given numerically.

4.2 Mesh

The geometry was split into cells so the governing equations can be solved numerically. The mesh should be fine around the heat sink surfaces and narrow ducts to ensure accurate results. The mesh sizing should be mentioned along with the number of cells.

The quality of the mesh should also be mentioned. If possible the mesh should be independent. The mesh was created in the report but wasn't given numerically.

4.3 Material Properties & Boundary Conditions

The heat sink was created using aluminium and air as the cooling medium. To ensure the CFD study can be replicated the exact material properties should be mentioned. Mention if temperature dependent properties were used. The inlet velocity was varied. The outlet pressure, inlet air velocity, heat input/imposed thermal conditions, wall conditions and turbulence model should be stated.

4.4 Solving & Post-processing

The flow and energy equations were solved using ANSYS Fluent. The main result looked at was the temperature. This can help identify where the hotter and cooler areas are. Velocity and pressure contours can help explain the airflow. To ensure the results have converged the residual, mass and energy balance and relevant results should be stable. The residual target wasn't mentioned.

4.5 Cases

Three simulations were carried out. The inlet air velocities were 1m/s, 2m/s and 3m/s. To ensure the comparison between the cases was fair the geometry, material properties, heat input, inlet temperature, outlet condition and solver should remain the same. Only the inlet velocity should be different.

5. Results

The following temperatures were found. The table shows the highest and lowest temperatures in Kelvin. However, there are statements within the report mentioning temperatures in Celsius. This cannot be explained. The following temperatures were found:

Inlet Velocity (m/s) Maximum Temperature Minimum Temperature Unit

1 309 306 K

2 305 302 K

3 302 299 K

Table 1: Temperatures from the project report. Should check Fluent result files. Other parts of the report mention temperatures around 299-310OC.

5.1 Trend

As shown in the table, the highest temperature is decreasing from 309 to 302. This is a difference of 7. The lowest temperature is also decreasing from 306 to 299. As seen from the table, the highest and lowest temperatures are lowest for the 3m/s case.

Overall this trend is expected as general heat transfer principles apply. Increasing the air velocity can improve the forced convection on the heated surface. However this does not show that the system will have good cooling performance for a real battery pack.

This should be interpreted with the same units, clear thermal load, comparable boundary conditions and evidence that the solution has converged.

5.2 Key Point

The temperatures throughout this report are inconsistent. The temperatures are given in the table in Kelvin. However, other parts of the report mention temperatures such as 309, 302 and 299 in Celsius. This is a big difference in temperatures. This paper will not convert any temperatures. The authors should open up the original ANSYS Fluent case/data files to see what unit the temperatures are in the contour legend. Update the table and discussion with the correct units. The report mentions that the maximum temperature was around 310OC. This does not directly correspond to the table. If it can be confirmed from the simulation output then this can be included.

5.3 Limitations

Not enough validated information was provided so the heat transfer rate, % improvement, average temperature, pressure drop, fan power and uncertainty couldn't be calculated. If possible include exact sizing of the geometry, heat input, inlet temperature, mesh statistics, solver models, residuals and any experimental measurements.

6. Discussion

This comparison suggests that the inlet air velocity is an important factor in the performance of the model. If everything remains the same a higher velocity will mean cooler air reaches the heat sink quicker improving the convective heat transfer. The 3m/s case had the lowest maximum temperature. Therefore if just looking at the temperatures the highest velocity would be the best option.

However, this shouldn't be the only factor to consider when deciding on the design. A higher velocity may cause pressure losses and increased electrical power requirements for the fan. The air may not flow evenly causing hot spots. Future comparisons should look at the maximum temperatures, temperature uniformity, pressure drop and estimated fan power. These results are numerical predictions and shouldn't be used as evidence that the battery will operate safely.

7. Conclusion

This study looked at an aluminium heat sink and air duct for potential battery cooling in an electric vehicle. Three cases were compared using the temperatures given in the project report. The maximum temperatures were 309, 305 and 302. The minimum temperatures were 306, 302 and 299. As shown in the table the 3m/s case had the lowest temperatures.

This suggests further investigation could be carried out on forced air cooling. This has shown how useful CFD can be to compare different operating conditions. However, there was a large inconsistency in the temperatures. The temperatures were given in the table in Kelvin but mentioned in the text. The temperature unit and maximum temperature should be checked against the original Fluent files. The design should be validated experimentally.

8. Future Scope

Check all the temperatures and units from the ANSYS Fluent case/data files.

Mention sizing from CAD, heat input, inlet temperature, outlet pressure, material properties, mesh statistics and solver settings.

Carry out mesh independence study. Mention any convergence checks, mass and energy balances.

Investigate further air velocities, fin height/thickness/spacing and duct design.

Calculate pressure drop and fan power to see the overall efficiency of the cooling system.

Create a prototype and compare the temperatures.

Expand the model to include multiple battery cells and investigate transient charging and discharging conditions.

References

[Volume] Buonomo, B., Manca, O., Menale, F., & Nardini, S. (2021). Numerical investigation on the thermal control of lithium batteries for electric cars using metal foams and phase change materials. Journal of Physics: Conference Series, IOP Publishing. [Article Number] [DOI]

[Volume] Guo, H., Li, J., Zhu, Z., & Lv, P. (2022). Numerical investigation on thermal performance of battery module with LCP-AFC cooling system applied in electric vehicle. Energy Science & Engineering, Wiley. [Issue] [Pages] [DOI]

[Volume] Fatukasi, S.O., & Bello-Ochende, T. (2022). Numerical development of effective cooling system for battery pack of electric vehicles. Materials Today: Proceedings, Elsevier. [Pages] [DOI]

[Volume] Al Shdaifat, M.Y., Zulkifli, R., Sopian, K., & Salih, A.A. (2023). Numerical simulation on heat transfer performance of a liquid cold plate for cooling of the electric vehicle battery. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 102(1). [Pages] [DOI]

[Volume] Sirikasemsuk, S., Wiriyasart, S., Prurapark, R., Naphon, N., & Naphon, P. (2025). Numerical analysis and testing of battery thermal cooling performance using ferrofluid through mini-channel heat sink units. Engineered Science. [Full Publication Details]

[Volume] Tayyab, R., Aziz, H., Imran, M., Waseem, Z., Anwar, M., & Khan, T.I. (2025). Thermal management of dry batteries for electric vehicles. Engineering Proceedings, MDPI. [Article Number] [DOI]

[Volume] Gu, J., Kim, H.-K., & Jang, S. (2025). Study of cooling performance of liquid cooled EV battery module according to the TIM compression ratio. International Journal of Automotive Technology, Springer Nature. [Pages] [DOI]

[Volume] Esen, D.Ö., & Shabaneh, S.A.Y.A. (2026). Integrated thermal management strategies for electric vehicles: A system-level review of battery cooling, heat pump technologies, PCM integration, and cold climate performance. Journal of Thermal Analysis and Calorimetry. [Published?][Full Publication Details]

Author Checklist

Check temperatures are in Kelvin or Celsius by checking the contour legend in Fluent. Ensure consistent throughout.

Update notes with correct Volume, Issue, Pages/Article Number and DOI.

Only include images of actual CAD/Mesh and ANSYS temperatures if it is your own simulation.

Include actual heat input, inlet temperature, mesh sizing/cells, turbulence model and convergence criteria.

Ensure all authors have checked the text and correct requirements have been met for research and citations from the institution.