ENS5146 Engineering Honour Thesis 2

Final Report

Computational Fluid Dynamics (CFD) Simulation of a Battery Thermal Management System Using Ansys Fluent

Syed Ismail Hussain Shah

Student 10502143

Bachelor of Engineering (Mechanical)Honours

Edith Cowan University

School of Engineering

Date of Submission: 27 October 2023

Supervisor: Dr Abdellah Shafieian Dastjerdi

Copyright and Access

Author Declaration

I certify that this thesis does not, to the best of my knowledge and belief:

(i) incorporate without acknowledgement any material previously submitted for a degree or diploma in any institution of higher education;
(ii) contain any material previously published or written by another person except where due reference is made in the text; or
(iii) contain any defamatory material.

Name: Syed Ismail Hussain Shah

Signature: Syed Ismail Hussain Shah Date: 27/10/2023

Abstract

In this study, a computational fluid dynamics (CFD) model to simulate the thermal performance of a battery is done with the help of Ansys fluent. Batteries have continued to grow in terms of significance in the contemporary world. Lithium-ion battery's thermal behaviour is significant for the battery's safety, degradation, and performance. Therefore, it requires both modelling and measurement. The study has noted that establishing a battery thermal software model, which engineers can use to simulate the heat transfer process, is an effective way of analysing any form of trade-offs that may exist regarding the parameter of battery design, evaluation of the battery's performance, as well as ensuring that control algorithms are implemented effectively. To achieve the objectives of this study, the various aspects that can be used to improve the battery's performance, the main focus was on CFD post-processing. This involved establishing models regarding the temperature contours and vector plots at heat flux of various levels (31W/m2 and 50 W/m2), temperature contours and vector plots at different diameters (0.5mm, 0.8mm, and 0.11mm) and at distant mass flow rates (16L/h, 21L/h, and 26L/h). The simulation has been verified in a controlled environment using the varied rate of discharge experiments. Infrared photographs of the power source cell were also acquired during discharging, and the resultant quantitative gradient temperatures were compared. The CFD models were validated by comparing the ambient temperature, current state of charge, and voltage curves to experimental data. The model estimates closely match the experimental facts. The issue in CFD battery calculations using an electrolytic approach is that many physical variables are challenging to discover. The features of the modelled battery's properties have been assumed and proven in this work; these can be beneficial for modelling batteries of a similar sort. As a result, the model created during this study can be utilised to forecast the variation in temperature of the LiPo cylindrical batteries and may be employed by battery designers and system architects.

Acknowledgement

In completing this final year thesis report, I express my sincere gratitude to the number of people who have helped me in making this journey possible.

First and foremost, I want to thank my supervisor, Dr Abdellah Shafieian Dastjerdi, and Co-supervisor Prashan Perera, who have been my dedicated supervisors throughout this project, for their advice, knowledge, and constant support. Their mentoring has greatly influenced this work.

I also want to thank my family (Father, Mother, Brother, Sister, Sister-in-law, and Nephews) for their support, love, and tolerance throughout my academic journey. They have provided excellent assistance.

Finally, I want to thank friends and fellow students for their support and companionship throughout this challenging but worthwhile journey.

Contents

1. Introduction

1.1 Introduction

Batteries are gradually replacing conventional energy sources, particularly in the transportation sector. In Australia, for instance, the demand for energy sources with less air pollution has increased significantly. As of February, this year, the number of electric vehicles in Australia almost doubled, from 44,000, recorded at the beginning of 2022, to more than 83,000 (VIsontay, 2023). This number is expected to increase to approximately 100,000 in the next three months, with 79% of these vehicles running purely on lithium batteries and 21% hybrid plug-ins (VIsontay, 2023). Currently, EVs and hybrids in Australia account for approximately 3.8% of all the new vehicles purchased in Australia, and the market share for these vehicles is substantial in the country's capital territories (VIsontay, 2023).

However, with EVs and hybrids becoming more popular, engineers must shift their attention to the safety and efficiency of the batteries used to run these vehicles. Currently, there are different batteries (i.e., lithium-ion batteries, nickel-metal batteries, lead-acid batteries, etc.), with each type of battery expected to produce a specific performance and safety when used. Due to their high power per mass compared to other electrical energy storage methods, battery packs are currently employed in most portable consumer gadgets, including cell phones and laptops (Morone, Cotton & Giudice, 2023). They also have superior efficiency, a high wattage ratio, outstanding energy efficiency, and minimal self-discharge. Although recycling most lithium-ion battery parts is possible, the expense of material recovery must be addressed for the sector. As shown in Figure 1, each battery type is characterized by its power and energy density.

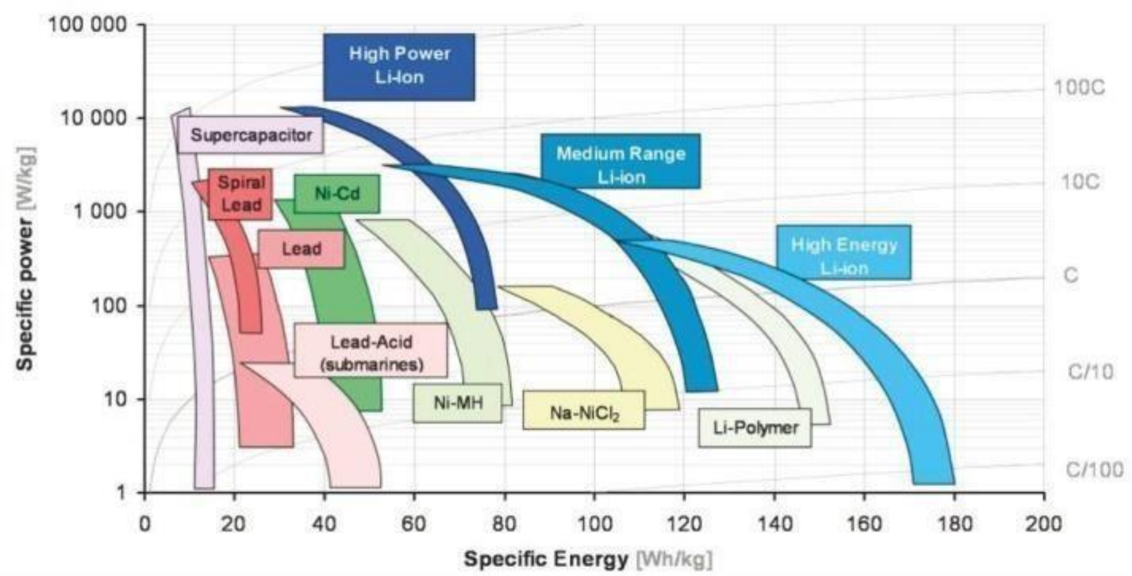


Figure 1 Power and energy density of each battery type (Olabi et al., 2022)

The temperature significantly impacts battery capacity and life; battery thermal management must be employed in hybrid, electric, and plug-in hybrid automobiles under real-world operating situations (Ni & Wang, 2020). In the past few years, manufacturers and their electric vehicle battery vendors have placed a greater emphasis on battery heat management, particularly in terms of lifespan and warranty expenses (Kim & Pesaran, 2007).

The idea presented in Figure 2 depicts the heat management modelling method of the battery at the National Renewable Energy Lab (NREL) (Kim & Pesaran, 2007). The NREL's batteries thermal control layout model considers cell features (e.g., form factor, chemistry, materials, and dimension), operating conditions (e.g., power load profiles from the vehicle and ambient temperatures), and module/pack cooling processes strategy (e.g., cooling fluid type and the rate flow of mass, the channel design of the coolant, and the duty cycle of temperature). To forecast the temperature response of the architecture, the model employs the inputs mentioned above for part and system evaluation (Kim & Sesaran, 2007). The potential design adjustments can then be reviewed to find the best solution while considering the price, size, mass, and upkeep difficulties.

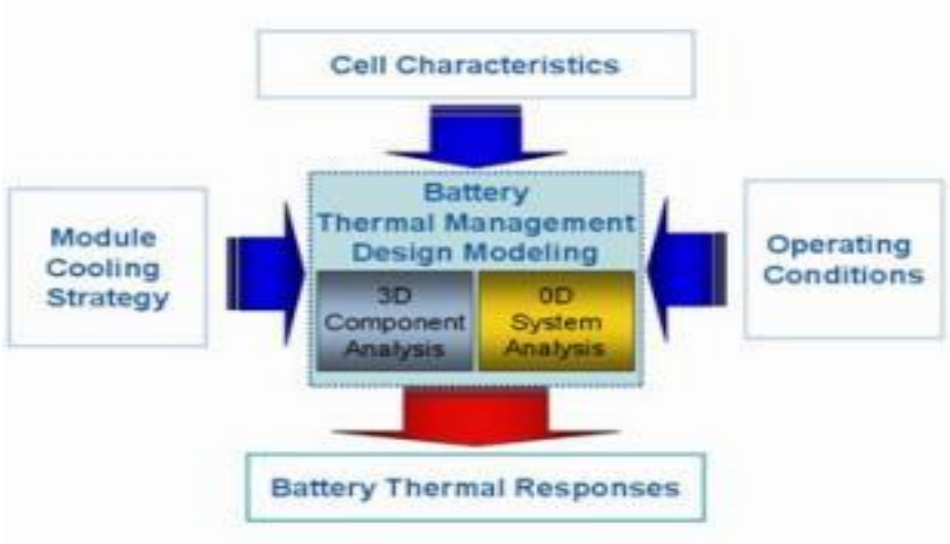


Figure 2 Battery thermal responses (Kim & Sesaran, 2007)

The current report examines viable heat management solutions. It is vital to assess system thermal responsiveness and responsiveness as an outcome of controlled system variables to identify an outstanding performance yet affordable cooling system. This study offers analyses and methodologies that engineers should consider when designing a vehicle's battery temperature management system (Kim & Pesaran, 2007).

1.2 Objectives

Batteries are essential in today's engineering field. At the same time, creating battery thermal software models, which can help engineers to simulate the process of heat transfer, can help them to effectively analyse any trade-offs in terms of the design parameter, evaluate the performance of the battery, as well as ensure that control algorithms have been effectively implemented (Bamrah et al., 2022). Considering this aspect, therefore, the main objectives of this project are:

1. The Ansys Fluent software is used to develop a computational fluid dynamics simulation of a battery thermal management system.
2. Review areas of the battery thermal management system that need to be improved to ensure the effectiveness of the process.
3. Examine future improvements in the battery thermal management system based on the results of the CFD simulation.

1.3 Significance

By controlling the temperature of batteries, a battery thermal management system ensures their safe and effective operation (Jindal et al., 2022). Low temperatures can cause a reduction in battery capacity and poorer charging/discharging performance, while high temperatures can hasten battery aging and pose safety issues. Especially in a world where many people are slowly adopting EVs and hybrid vehicles, capturing the thermal behaviour of batteries is very important (Jindal et al., 2022). At the same time, understanding the various methods used to capture the thermal behaviour of batteries is crucial because it can help one identify issues with the method, which may prevent engineers from accurately capturing the thermal behaviour of batteries. While this project is mainly objectified at establishing a CFD simulation of a BTMS using Ansys Fluent, therefore, the final product of this project will also involve literature on various issues associated with BTMS and how these issues can be improved in the future to make the whole process a success for engineers. Learning more about battery thermal management is particularly important because it will help establish studies on which future scholars and engineers will base their studies. Kim and Pesaran (2007) also noted that models capable of predicting batteries' thermal behaviour are likely to shorten the process of development, which is required to improve the design system of batteries. Therefore, the thermal management of the battery is essential mainly because it helps achieve performance, as well as helps to extend the battery life, especially for hybrid and electric cars (Shah et al., 2023). This study will prove to be very significant for engineers currently trying to identify different ways of managing the thermal aspects of batteries.

1.4 Report Organisation

The report is divided into five different chapters named as follows:

- Introduction
- Background study or Literature review
- Proposed approach
- Preliminary results and discussion

- Conclusion

Chapter 1 provides the project introduction along with objectives and significance. This part of the report briefly introduces the project, along with its expected benefits of doing this project and some aims as well.

Chapter 2 presents the background of this project, and this background research part of the report also shows how much work has been done on this topic which helped the successful completion of this project.

Chapter 3 examines the proposed approach, which will be used to do the project's different parts step-by-step.

Chapter 4 addresses the results and discussion, with some line graphs and explanations.

Chapter 5 concludes the report with the conclusion and the list of references used in the report.

2. Background

2.1 General Background

The motivation behind this project is to develop knowledge regarding the battery thermal management system (BTMS) with the aid of computational fluid dynamics (CFD) simulation. A BTMS regulates the battery's operational temperature by supplying heat during extreme cold or dispersing heat during extreme heat. Engineers adjust battery temperature in these systems using passive, active, or hybrid transfer techniques for transferring heat. In active solutions, the battery's temperature is often changed by a fan or pump pumping a working fluid, such as water, air, or another liquid (Bamrah et al., 2022). A passive approach involves transferring heat away from the battery using either heat sinks or pipelines made of thermally conductive materials. Key design elements of both passive and active solutions are combined in a hybrid solution (Jindal et al., 2022).

Moreover, today's communities are looking into alternative energy sources due to the excessive use of fossil fuels and oil products and the increased demand for efficient energy systems (Olabi et al., 2023). Due to the continued reliance on vehicles that run on fuels like gasoline and diesel for day-to-day operation, dangerous by-product emissions have reached alarming heights. 3 to 5% of the vehicles use alternative fuels like ethanol and natural gas, but about 90% run on petroleum products. The last ten years have seen increased public awareness of and sensitivity to the need for creative solutions for producing clean energy among citizens, the government, businesspeople, and scientists (Rosencrance, 2021).

As another alternative to traditional fuels, battery-operated cars labelled Electric Vehicles (EVs) have become increasingly popular in the last several years. According to projections, there will be over 18.7 million electric vehicles (EVs) on the road by 2030, about 15 times more than in 2017 (Olabi et al., 2023).

However, while batteries have gained significance as alternative energy sources, knowledge of their operation regarding safety and efficiency has gained significance in the engineering field. Another vital point to note is that with aid tools such as the Ansys Fluent software, CFD simulation can be used to learn how BTMS works and identify whether the process may need other improvements to make it more effective in engineering.

2.2 Increasing Role of CFD in the Contemporary World

The role of CFD in design is increasing as more people continue to embrace the use of hybrid and electric cars and other things that require batteries. As thermal issues and limits become more prevalent in semiconductor design, computational fluid dynamics technologies have begun to appeal as a means to simulate, analyse, anticipate, and, ideally, prevent thermal problems from occurring (AKG of America, 2018). Everything from conditioning an entire board to conditioning a chip using a fan and a heat sink relies on air circulation for cooling or, in certain extreme circumstances, liquid flow. Fluids keep the chip cool in all instances. It is impossible to say the amount of energy the device will produce, but the temperature it can attain depends on its cooling atmosphere. CFD is required to predict the fans and radiators and if two hot parts placed close to each other will behave differently than if they are far apart (AKG of America, 2018).

CFD is a crucial component in large-scale simulation. It takes work to eyeball it. CFD is excellent for modelling since there are excellent models for mathematics. Wind tunnels have historically been utilised to design aeroplanes (CFD Flow Engineering, 2023). You constructed something, tested it, tinkered with it, and then recreated it. There is now a push toward simulations to detect problems earlier. A hypersonic missile, for example, is extremely difficult to test. Thus, it must be simulated.

Furthermore, CFD is widely used in the automobile industry, alongside aerospace. In the instance of electric automobiles, the electric motors are so quieter than the combustion engine that the circulation over the rear-view mirrors is one of the most significant noise sources (AKG of America, 2018). CFD may be used to build the mirrors so that airflow disturbance over the mirrors is minimised. CFD may also be employed in creating vehicles with less drag to increase the distance per charge of electric vehicles. Aside from price, the most important consideration when purchasing a car powered by electricity is range (AKG of America, 2018). This concept is well represented in Figure 3.

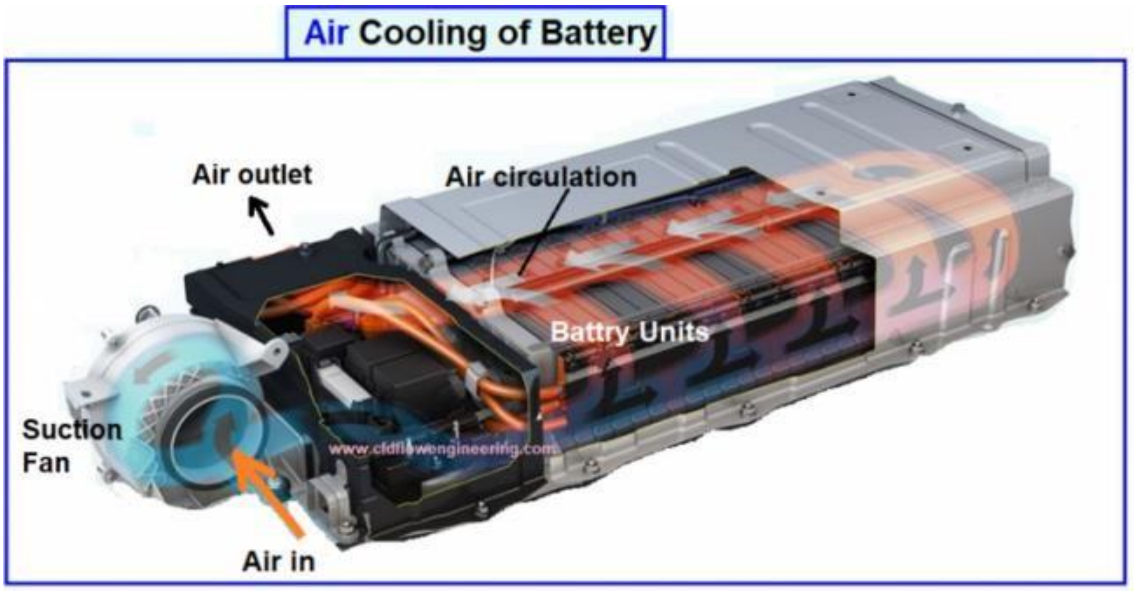


Figure 3 The use of CFD to cool automobile batteries (AKG of America, 2018).

CFD is also used in the cooling mechanisms within vehicles. The thermal issue with an internal combustion engine occurs while you are stuck in traffic since one must cool the engine and have various cooling systems. The difficulty for an EV comes during charging (AKG of America, 2018). So CFD may be utilised in aerodynamic and cooling systems to model water flow and how it can cool off systems. It could be some converter. For batteries, new concepts will emerge in which you have a battery pack and spray water from the top onto the battery while employing a supercharger (Teschler, 2017).

2.3 The use of CFD in Advanced Packaging Usage

CFD technique is likewise gaining popularity as Moore's Law slows. Instead of expanding, semiconductor manufacturers are embracing new and increasingly diverse architectures, as many die in a high-end package. Stacking die within a 2.5D or 3D packaging significantly raises the difficulty of thermal dissolution, which can result in issues ranging from rapid and unequal aging to stress fractures produced by thermal mismatch between various parts and materials. Thermally induced chip-substrate package interconnections can be significant, and proper control is critical to guaranteeing acceptable reliability (Teschler, 2017).

Depending on the performance, the die can be composed of various base materials, such as silicon, semiconductor, the metal germanium, germanium arsenide, etc. The interposer can be built of silicon, ceramics, glass, or organic materials. These substances grow at various rates when heated due to their varied coefficient of thermal expansion. Semiconductors are extremely rigid substances that do not easily bend (AKG of America, 2018). While the framework bends, the substances sandwiched between the layers encounter shear, which can cause electrical connections to fail. The connector to the board is also affected by overall packaging expansion and bending.

Systems integrators and OSATs must understand the 3D distribution of temperatures within the package architecture inside the application context to ensure dependability. The temperature distribution throughout the package is determined by where and how heat is dissipated from the packaging into the board, as well as any associated cooling solution such as a heat sink or compressor block. Because these heat flow pathways interact, any change in design affects heat flow everywhere (Teschler, 2017). CFD is required to forecast heat movement across the system and, thus, the package construction. CFD primarily allows for precise thermal simulation since it accurately forecasts surface heat transfer rates due to circulation and thermal radiation.

Over time, the emphasis has shifted to the packing structure, from box to PCB/board, IC pack, and finally to the die level. Accuracy demands have also increased. Accurate thermal simulations for the elements, PCB, and any component-mounted heat propagators or heatsinks are required to fully model transmission, custom, and radiant heat (AKG of America, 2018). Furthermore, CFD is being used in areas that were not considered before. "CFD is an essential component of the evaluation chain when modelling thermal effects." How can you get rid of the heat? You must be able to simulate fluid. Air is a type of fluid. You must be able to simulate passive vs active cooling approaches. A forced-air fan may cool base station equipment, such as the baseband or control plane unit. One has a way of getting the heat out, and your method should leave enough room for the unit to work in Siberia or Death Valley. People can use CFD to look at stuff like this. It combines thermal analysis with optimization of the architecture of these systems. This concept is well visualised in Figure 4.

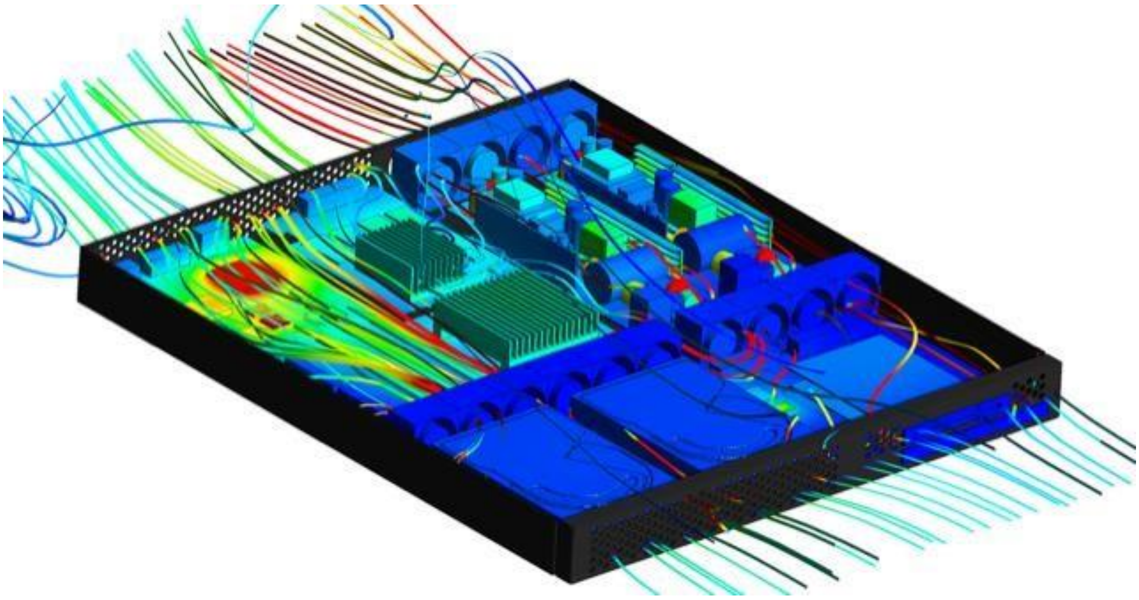


Figure 4 Understanding how heat is critical in a system (AKG of America, 2018).

2.4 The Use of CFD to Examine BTMS

When used in various components and not only EVs, the performance and safety of batteries are very important (Bamrah et al., 2022). Various techniques and mechanisms have been developed to help with this aspect.

The computational fluid dynamics (CFD) evaluation of the circulation is carried out within the BTMS utilizing ANSYS Fluent. The examination of the circulation of air throughout the battery modules can provide better insight into modifying the cell packing design and the placement of either passive or active thermal control devices (Bamrah et al., 2022).

The virtual environment will educate students about how BTMS operates regarding battery safety, health, and effectiveness. In addition to some of the issues connected with BTMS, I have studied how to perform a step-by-step Battery Thermal Management System (BTMS) simulation. Figure 5 below shows the overview of what I have explained in the above few lines. The below figure shows that, first of all, a battery pack will be designed in the software (Teschler, 2017). In this case, as a software, I am using Ansys fluent, so the design modeler of Ansys fluent will be used to create the battery pack design. After that, we will proceed with the mesh generation, and for that purpose, Fluent (With Fluent meshing) will be selected because that option has several steps. Following that number of steps can make the meshing results more precise in a short duration. Then, CFD analysis comes into consideration, which mainly relies on numerical values, where velocity, mass, temperature, and pressure values are adjusted if required, and setting the physics also comes under CFD analysis. Lastly, a simulation will run, and desired results will be achieved (Mutschler, 2022). If they are acceptable or within a reasonable range, we will leave that, or if they are in an acceptable range means the plots have continuity, and then some modification will be done.

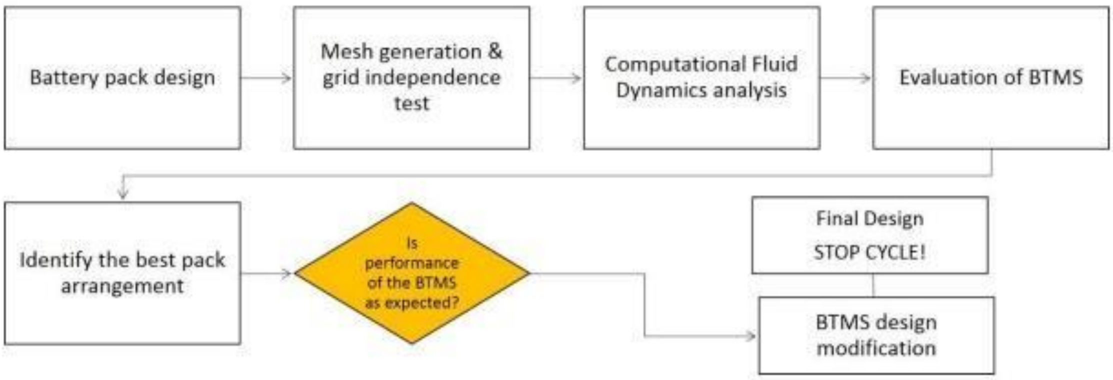


Figure 5 BTMS design modification (Mutschler, 2022).

2.4.1 Computational Fluid Dynamics (CFD) Simulation

Mathematically forecasting actual fluid flow by utilizing computers to solve governing equations is known as computational fluid dynamics (CFD) (Tsavlidis, 2023). Aerodynamics is critical to a design's overall performance when an engineer creates a new product, such as a winning race vehicle for the upcoming racing season. However, aerodynamic effectiveness is difficult to measure in the concept stage. Traditionally, the only option for an engineer to improve his/her concept is to perform testing procedures on product prototypes. The field of CFD has developed into a widely used technique for forecasting real-world physics due to the rise of technology and continuously increasing computational capacity (due to Moore's law) (Tsavlidis, 2023).

Researchers have tried to examine areas where CFD has been used effectively. In one of the studies, for instance, Nowak et al. (2003) examined how CFD is used to simulate airflow and the disposal of aerosol in human lungs. In each model, particle trajectories were generated by solving the particle's motion equations for the predictable part of particle displacement and then adding a stochastic Brownian factor at each step. Particle trapping on the external walls was observed, and each generation's trapping locations were noted (Nowak et al., 2003). Ultimately, the researchers concluded that more than an idealized model based on the Weibel dimensions is required to predict particle flow.

Moraveji and Toghraie (2017) did another study to investigate how CFD may be used to investigate the characteristics of heat transfer and flow in a vortex tube according to different parameters. Moraveji and Toghraie (2017) determine via the information collected in their research that the speed of movement rate from the cold connection increases with weight while the mass that moves flow rate via the cold, as well as hot bridges, rises modestly with the overall length of the vortex tube. Furthermore, as the percentage of inlets grew, so did the temperature variations at both channels, while the radius of the cold exit increased (Moraveji & Toghraie, 2017). The gas's temperature when exiting the outlet is also significantly higher than the temperature changes at the hot and cold channels in comparison to the particular instance where more inlets with smaller diameters are used.

2.4.2 Battery Thermal Management System

Similar to CFD, BTMS has also gained significance in the engineering field, especially regarding the use of this system to examine the safety and effectiveness of batteries used in most devices. Researchers have mainly done several reviews of BTMS to examine its effectiveness. Lin et al. (2021), for example, said that an effective BTMS is required to keep the battery's temperature below the acceptable range and reduce the temperature differential between cells. The highest temperature rises and the most significant temperature difference of the battery pack itself are the two primary variables utilized to evaluate the efficiency of the BTMS. To maintain the best results and prolong the life of an electrical battery, the outside temperature of each cell in a power battery has to be kept between a defined range of 20 °C and 45 °C, with a significant temperature difference between cells being less than five °C The temperature profile across an operating cell can be described using Gregory's latest key indicator of the cooling coefficient, which he proposed (Lin et al., 2021).

In another study, Olabi et al. (2022) examined the recent progress made in the development of BTMS. The researchers argue that thermal problems may significantly impact the battery's performance and lifespan. In order to build a reliable and effective system that is not adversely affected by changes in internal and external temperature, a proper BTMS is required (Olabi et al., 2023). Moreover, BTMSs are required to lower costs while improving the battery's safety, cycle life, and efficiency. One of the main reasons why BTMS is regarded with significance is the influence that operating temperature has on the performance of batteries. This aspect is shown in Figure 6.

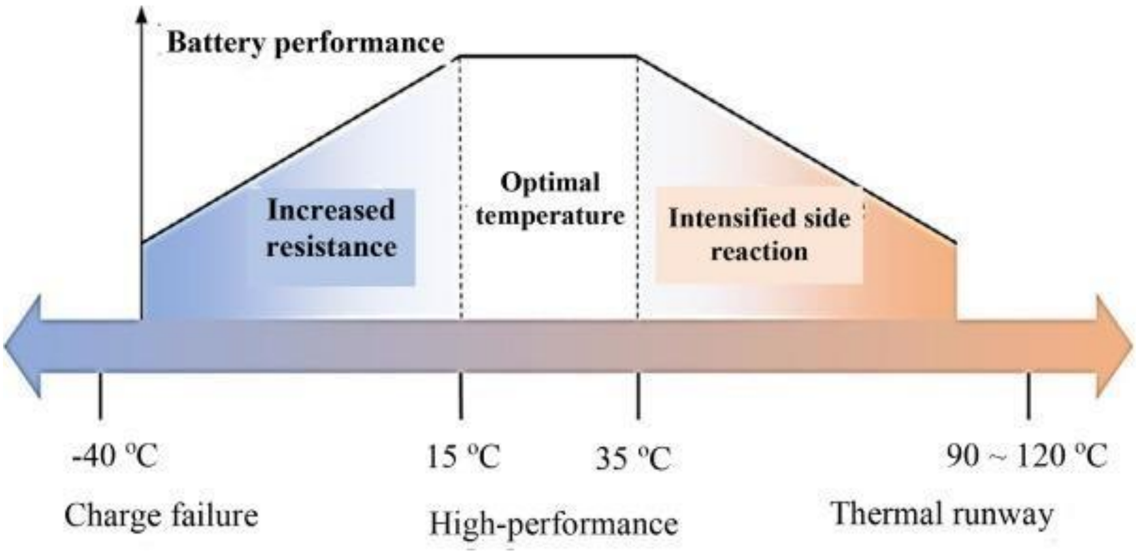


Figure 6 The range of temperature in batteries (Katoch & Eswaramoorthy, 2020)

Batteries' safety, efficiency, and lifespan are all affected when they operate in temperatures outside their working ranges. However, terrible environmental circumstances, such as exceptionally or exceedingly low temperatures, are unavoidable in actual applications. Under these unfavourable circumstances, the battery's efficiency will dramatically decline due to improper storage or operation temperature (Nowak et al., 2003). The battery could sometimes have a thermal runaway (TR). This section will examine the damaging effects of low temperatures on batteries to comprehend the value of BTMS. Unexpected thermal characteristics are caused by low temperature, non - uniformity, high temperature, and high-performance degradation (Katoch & Eswaramoorthy, 2020).

According to previous research, the duties of this position will be utilized to gain additional information about BTMS. The data gathered will also be utilized to forecast future BTMS improvements to improve and render the procedure more effective (Lombrado et al., 2021). The graph below displays the articles over the previous 20 years and illustrates that BTMS has become increasingly important from 2016-2020. Similar to CFD, BTMS has also gained significance in the engineering field, especially regarding the use of this system to examine the safety and effectiveness of batteries used in most devices. Researchers have mainly done several reviews of BTMS to examine its effectiveness. For instance, Lin et al. (2021) noted that maintaining battery temperature within the appropriate range and reducing the temperature difference between cells require an effective BTMS. The maximum temperature increase and the highest temperature differential of the battery pack are the two key factors used to assess the BTMS's performance. The cells' temperature in a power battery must be kept within a specific range between 20 °C and 45 °C, and the most significant temperature differential between cells must be less than five °C to maintain optimal results and extend the life of a power battery. The temperature profile across an operating cell can be described using Gregory's latest key indicator of the cooling coefficient, which he proposed (Lin et al., 2021).

In another study, Olabi et al. (2022) examined the recent progress made in the development of BTMS. The researchers argue that thermal problems may significantly impact the battery's performance and lifespan. In order to build a reliable and effective system that is not adversely affected by changes in internal and external temperature, a proper BTMS is required (Olabi et al., 2023). Moreover, BTMSs are required to lower costs while improving the battery's safety, cycle life, and efficiency. One of the main reasons why BTMS is regarded with significance is the influence that operating temperature has on the performance of batteries. This aspect is shown in Figure 7.

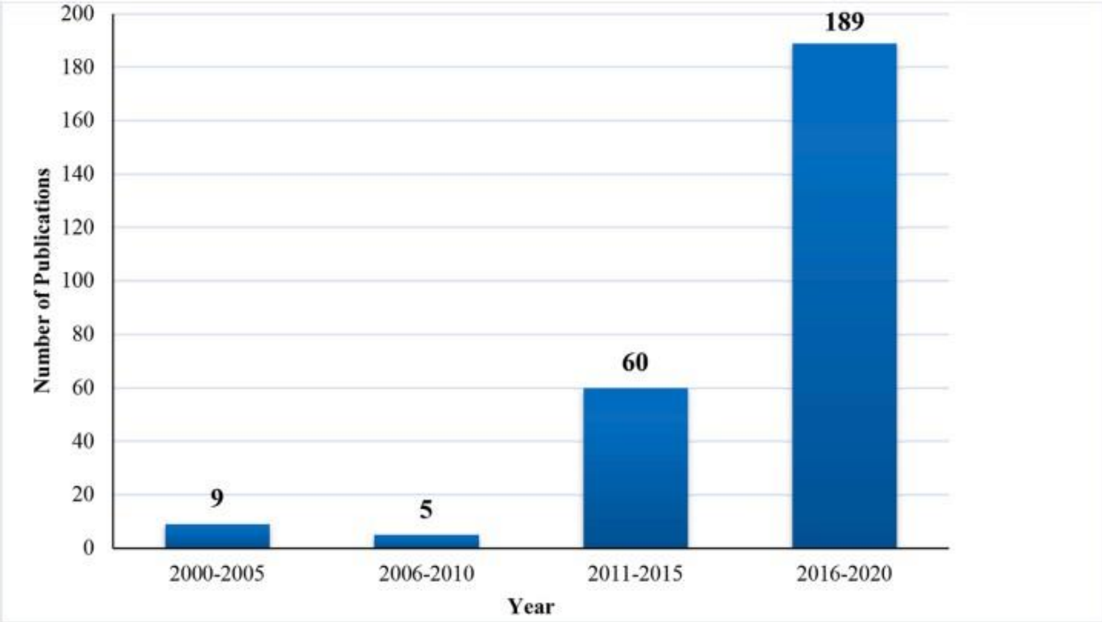


Figure 7 Year-wise statistics of the number of papers reviewed on BTMS (Olabi et al., 2023).

3. Proposed Approach

The proposed approach is the most essential step in achieving the desired objectives of the project. The way any problem is solved matters in getting the desired outcomes. This project mainly deals with the simulation in Ansys fluent, but simulation is the last step, so several steps should be followed before the simulation. This is called the proposed approach. The proposed approach for this project starts with the literature review on battery thermal management systems and then learning about the software Ansys Fluent. Most of the work is on the Ansys Fluent software, which involves some steps listed below.

3.1 Literature Review

BTMS regulate battery temperature using passive, active, or hybrid transfer techniques, with increased public awareness of alternative energy sources. CFD has become increasingly important in design as more people embrace hybrid and electric cars and other things that require batteries. It is used to simulate, analyse, anticipate, and prevent thermal problems from occurring. CFD is used in large-scale simulation to detect problems earlier, such as the F-35 combat jet and electric automobiles, which are quieter than combustion engines. CFD can be used to build mirrors and create vehicles with less drag and excellent automobile batteries. CFD technique is becoming increasingly popular as Moore's Law slows, resulting in thermal dissolution issues and interconnections between chips and substrates. In order to combine thermal analysis with system architecture optimization, CFD is a crucial link in the evaluation chain when modelling thermal impacts. BTMS simulations offer insight into how BTMS functions with regard to battery health, safety, efficiency, and some of the difficulties that come with BTMS. Due to Moore's law, computational fluid dynamics (CFD) is a commonly used method for predicting real-world physics. It has been used to model airflow and the removal of aerosol from human lungs. In order to maintain optimal results and limit the temperature difference between cells, BTMS is crucial for evaluating the safety and efficacy of batteries. Examining the effectiveness and safety of batteries using BTMS has grown in significance in the engineering profession. BTMSs are essential for maintaining battery temperature within the appropriate range and reducing temperature differences between cells to extend the life of a power battery.

3.2 Ansys Fluent

Software for computational fluid dynamics called Ansys Fluent is mainly used for modelling fluid flow and heat transfer. With the aid of this CFD program, fluid-structure Multiphysics interactions and various fluid processes may be modelled and simulated (MR-CFD, 2022). Additionally, ANSYS Fluent includes many physical modellings features necessary for industrial applications, including fluid flow, heat transfer, turbulence, and reactions.

Many businesses and academics worldwide use ANSYS Fluent as their CFD software because of its unrivalled accuracy and reach. One may always be sure of their solution because it was created by famous engineers and experts worldwide, allowing them to generate better quality goods faster and with less risk and time to market. In summary, ANSYS Fluent is a very sophisticated CFD tool that meets the needs of every user.

The battery thermal management system simulation in Ansys Fluent comprises several processes.

- Geometry Modelling

- Fluent (with Fluent Meshing)
- Numerical Analysis
- Post Processing

3.3 Design Modeler

The ANSYS Design-Modeler tool, a customizable based on features solid modeller, is an easy and quick approach to start making 2D drawings, building 3D components, or uploading 3D computer-aided design (CAD) models for engineering evaluation pre-processing. Geometry modelling refers to using the ANSYS fluid workstation to create 2D or 3D representations which need extra computation to transfer heat or movement of fluids. The "geometry" provides a CAD design environment and capabilities comparable to Autodesk Inventor and SOLIDWORKS. Additionally, geometry for ANSYS Fluent may be imported from different CAD programs. Geometries from other programs must be imported as .stp files (ANSYS, 2012).

3.4 Geometry Modelling

Geometry Modelling is done in the design modeler of Ansys Fluent software. My project's geometry involves a 3D rectangular box with dimensions of 180 mm in length, breadth and 90mm in height. The geometry also includes five cylindrical batteries with a height of 65mm, an interior circle dimension of 18mm, and an outer circle of 24mm with equal spacing between them, as shown in Figure 8. Extruded boss is the only feature used in designing the below geometry, as shown in Figure 8. The right plane is being used, and from the different sketch tools, lines and circle options have been selected for creating the geometry. Named options are essential in ANSYS Mechanical studies. You can specify your named selections on geometry and utilize them in subsequent boundary condition formulations in ANSYS Mechanical, ANSYS Fluent, etc. It is a Handy tool for establishing boundary constraints in different ANSYS interfaces.

The below geometry shown in Figure 8 looks like a rectangular box named fluid domain.

That rectangular box has 6 faces; out of these 6 faces, 5 are named F1, F2, F3, F4 & F5. The only remaining face, which has some distance from the batteries, is called an outlet. Similarly, the names of the 5 batteries are B1, B2, B3, B4, and B5. The outer surface around the boundary of batteries is named BOI 1, BOI2, BOI 3, BOI 4, and BOI 5.

Lastly, the Boolean feature has been used, and it is to pick different solids to subtract or add using the suppress solid option. Furthermore, under the details of the Boolean option, preserved tool bodies have been selected. NO means they are hollow from the inside because we also want the internal surfaces of the batteries to mesh.

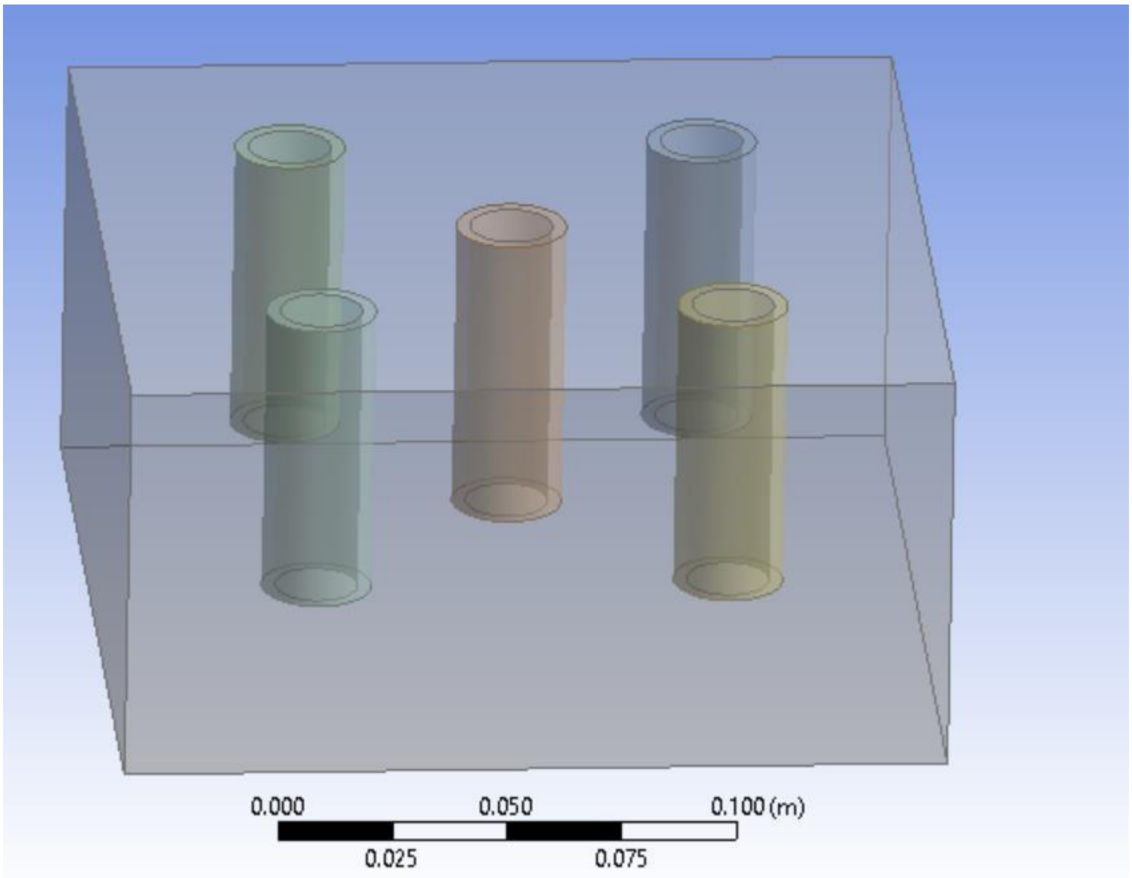


Figure 8 Geometry in design modeler

3.5 Fluent (with Fluent Meshing)

Fluent with Fluent meshing has been approached from several different options in the workbench of Ansys fluent. The reason for choosing this option is that it provides the user to work in different workflows according to the requirements. Moreover, Fluent meshing also benefits the user by adding local sizing and improving the mesh quality automatically if required. Furthermore, there are some other advantages discussed throughout the meshing discussion.

3.5.1 Meshing

Meshing has another name, mesh creation, which generates a 2D or 3D dimensions grid that helps differentiate complex geometries. (Simu Tech Group, 2023).

Meshing in Ansys is indicated as a standard tool in meshing complicated parts regardless of workflow type and shape of the geometry. Furthermore, Ansys meshing has some advanced tools that help in meshing CFD and FEA solutions with precise results in a short duration.

3.5.2 Importance of Meshing

Meshing is essential in getting a good simulation result in engineering measures. Precision in simulation is highly dependent on high-quality mesh, which should be considered in the meshing part of Ansys fluent (Simu Tech Group, 2023). The simulation results in precision, unity, and performance are directly linked with the meshing quality. If the mesh quality is good, getting a simulation with precise results would be easy. Getting accurate results in simulation helps one compare the experimental and actual results, but this only happens if the mesh quality is in the acceptable range.

Mesh elements enable the solution of equations that govern on reliably formed and theoretically defined surfaces. The problems addressed in these models often must be completed with differential problems. Because these computations are iterative, solving them by hand is impractical; hence computational techniques like CFD and FEA are used (Simu Tech Group, 2023).

3.5.3 Water-tight Geometry

The waterproof design was selected for the project's computational fluid dynamics (CFD) simulation for numerous reasons. Working with a sealed and waterproof geometrical design is critical in CFD simulation. This ensures that the fluid's region is verified for breaches or holes, and the flow behaviour inside that sealed fluid area is predicted (Ansys Innovation Space, 2023).

Another reason to choose the Watertight design is the simulation's perimeter condition. We need to verify that the configuration is watertight to assign the suitable inlet, outlet, barriers, mass circulation rate, and other choices to the correct geometry area.

Mesh integrity may also be employed to select watertight geometry (Ansys Innovation Space, 2023). Watertight geometry is characterised by narrow and precisely defined limits, ensuring that the mesh structure accurately fits the waterproof geometry's limits and surfaces. As a consequence, an excellent mesh is generated, which improves the accuracy of the simulation's result. Finally, before proceeding to the evaluation step, it will indicate any faults in the geometry's creation procedure or any geometry's shape discrepancies to guarantee that the simulation's findings are accurate (Ansys Innovation Space, 2023).

3.5.4 Add Local Sizing

In the add local sizing, all the BOIs have been selected that are defined in the design modeller. The growth rate selected was 1.2, with a target mesh size of 10mm. Moreover, in the add local sizing, the curvature option has been selected for all battery boundaries with a growth rate of 1.2, local minimum and maximum sizes of 2mm and 30mm and a normal curvature angle of 9 degrees. Moreover, the proximity option has been chosen for battery walls which are named as W1, W2 and W3 with growth rate of 1.2mm, local minimum and maximum size of 1 and 5mm. For the geometry edges, they are selected as edge size with a target mesh size of 1mm. Furthermore, Lastly, on meshing on the geometry's faces, the face size option has been applied on all 6 faces with a growth rate of 1.2 and a target size of 10mm. Lastly, it is always a best practice to employ the local control for refining the mesh for the selected part of the geometry to get the best quality mesh and refine the mesh in that selected part.

BOIs are closed 3D bodies that dictate the maximum cell size of the cell region that falls within their boundaries. BOIs also help define the region where the refined mesh is required, and they are not part of the final computational model. Use the Body of Influence (BOI) function of ANSYS Fluent to customize the mesh quality and precision in specific regions of interest. It allows users to fine-tune or coarse-tune the mesh, giving them versatility in collecting flow features, controlling accuracy, and maximizing computational resources. The BOI must accurately model complicated flow processes and appreciates localized impacts within the overall domain (CFD Ninja, 2020).

3.5.5 Geometry Description

Geometry description defines computational models by specifying the type of geometry, executing necessary sub-task, and defining or creating regions required for generating volume mesh. Furthermore, the fluid-fluid boundary is defined as the interface between multiple fluid regions, and this option has been selected Yes. This means all the fluid-fluid boundaries will change from wall to internal, and fluid will flow across regions but named selections that include wall are not converted to internal boundaries. In the Geometry I am working on, there is no inlet, and all the faces, including the boundary of the battery, consider walls. There is only one outlet that has a boundary type of pressure outlet.

3.5.6 Add Boundary Layers

In the boundary layer options, the offset method chooses to be the last ratio, with 10 layers, a first height layer of 0.1, and a growth rate of 1.2, and this boundary layer option only applies to the fluid region of the geometry. Furthermore, the number of layers represents the number of boundary layers on the surface of the geometry, and this number of layers is decided based on the level of accuracy required. Add in and grow on options define the location of boundary layer mesh generation. The Add-in option specifies the region where one would like to add a boundary layer. On the other hand, the grow-on option is used to specify the surfaces on which a boundary layer needs to be added within that region.

The transition ratio is the ratio of the height of the mesh in the last boundary layer, and the first cell in the volume fill. At the same time, the growth rate is the ratio of the thickness of the following boundary cell to the previous boundary layer cell.

3.5.7 Volume Mesh

Volume mesh generation represents a computational model using many discrete volumes or cells within which the governing equations are solved. Extreme care should be taken in generating a volume mesh because it significantly impacts accuracy, convergence, and simulation time. The fluent solver is selected to get precise volume mesh with a poly-hexacore fill, peel layer of 0, and minimum and maximum cell lengths of 2 and 64mm. Some advanced options have also looked like quality improvement limit, polyhedral mesh angle etc.

Furthermore, the benefit of using poly-hexacore is that it reduces the overall mesh count, better gradient approximation, and lower numerical diffusion. Maximum and minimum cell length is defined as the size of the largest and smallest cell in the domain. Lastly, the global boundary layer setting has selected yes with a gap factor of 0.25, minimum and maximum aspect ratio of 1mm and 26mm. After setting all the parameters, the volume mesh is generated with an orthogonal quality of 0.09. 427586 cells were created in 0.48 minutes, with 1891510 faces, 1091510 nodes and partitions of 4. In a Watertight geometry workflow, the default quality measure is orthogonal quality. Orthogonal quality measures alignment between the vectors normal to the cell faces and the vectors connecting cell centroids with face centroids. The orthogonal quality ranges from 0(Poor) to 1(Perfect). However, it is highly recommended to maintain it above 0.1.

Fluent meshing also has the option to improve the mesh quality if it is below 0.1. To improve the mesh quality, right-click on the Volume Mesh task, select insert new task, then select improve volume mesh; users then put the value above 0.1 to get a minimum acceptable orthogonal quantity mesh.

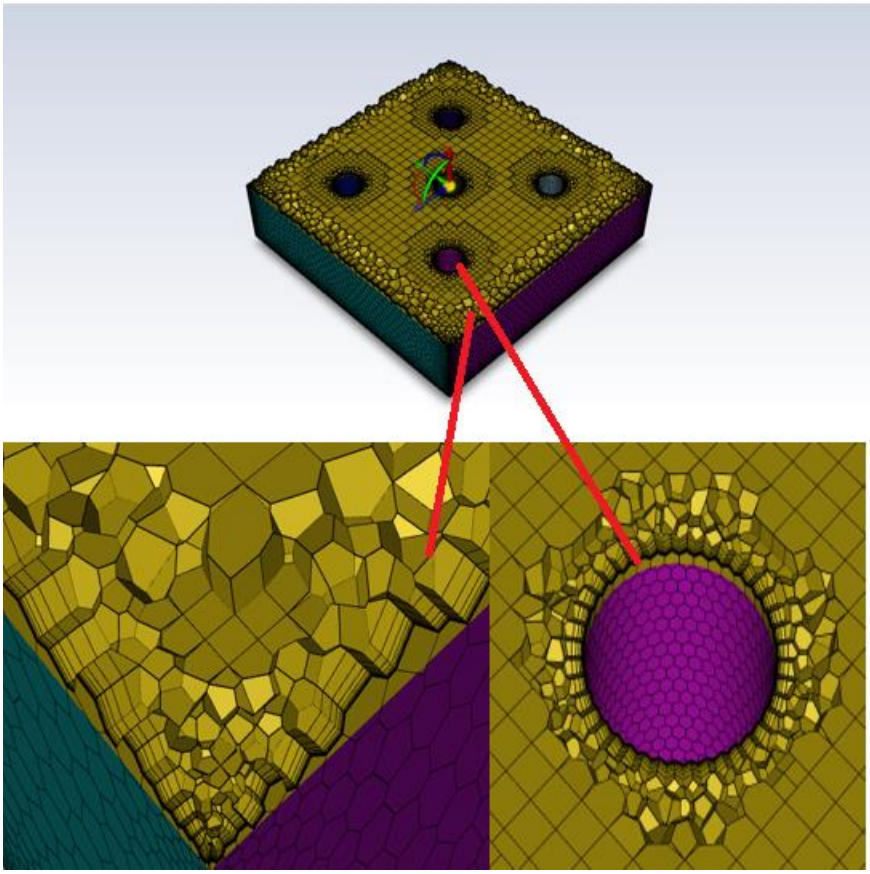


Figure 9 Volume Mesh

3.6 Physics Setup

The physics tab is equipped with many options to effectively define the problem which included specifying the type of

- Solver
- Relevant Models
- Material properties
- Cell zone conditions and boundary conditions
- Solution

3.6.1 Solver

The Pressure-based solution was chosen since it primarily works with indestructible and less compressible flows (Abderrahim, 2021). The injected material we use in our project is air and water vapour. This makes a pressure-based a good solver and a strong candidate for this project. Moreover, for the time option, time is selected as transient. Velocity formulation is selected as Absolute because the fluid is not moving in most parts of the domain and velocity is smaller in that frame. Therefore, it will decrease the numerical diffusion in the solution which helps the user to get more solution. Gravity option has been selected tick and the gravitational acceleration value for Y has been put -9.8m/s2.

From the solver time option, I selected transient because the flow of mixture (water vapour and air) in not constant. The transient time solver records these changes in temperature, pressure, and velocity and determines how these parameters advance at each time step. Another advantage of employing a transient time solver is that it breaks down the time domain into manageable time increments. By doing this, the solver computes the system state at the conclusion of each time step based on the previous state and also includes external influences like boundary conditions.

3.6.2 Physical Models

Modelling is divided into different parts. Firstly, from the Multiphase, the Volume of Fluid has been selected with a formulation implicit, volume fraction cut-off value of 1e^6 Number of Eulerian phases has been selected to 2. One is the air phase, and the second Phase is Water vapour. The Volume of Fluid (VOF) is chosen because it is well-suited for simulating free surface flows and interfaces. It primarily deals with sharp flows and provides accurate and detailed information. In addition, the fluids included in the VOF model maintain one set of velocity formulas, and the volumetric fraction of each liquid within each computing cell is tracked across the domain (Eghbalzadeh & Javan, 2012). The formula for energy has been set to On. The energy equation transfers heat between different system parts, including convention, conduction, and radiation. Considering the heat transfer between solid components like walls and batteries and the fluid domain is essential. Enable energy equation also include setup,

- Material thermal properties
- Thermal boundary conditions at the inlets and walls.

The reason for choosing the k-omega SST turbulence is because the model uses a k-formulation in the inner parts of the boundary layer, making it directly usable down to the wall through the viscous sub-layer and because the SST formulation also switches to a k- omega behaviour in the free-stream, it is more effective for this simulation. It provides more accurate results compared to other models. Another reason is that SST k-omega turbulent simulation is a two-equation eddy-viscosity simulation best suited for near-wall flow regions with unfavourable pressure gradients.

3.6.2.1 Discrete Phase Model (DPM)

Multiphase flows include the discrete phase model. The discrete phase model is used to examine the particle behaviour in terms of lagrangian and discrete points of view. Lagrangian and Eulerian views are different based on their flow behaviour. Lagrangian flow is studied based on tracking a particulate flow's particle. In contrast, the behaviour of the fluid in the Eulerian view is based on the notion that the fluid flow route contains a limited volume element.

Furthermore, water vapour is the primary phase in the project I am working on, so it is treated as a continuous phase. Air is a secondary phase that will interact with the continuous phase, water vapour. DPM iteration interval has been set to 10, tracking parameters like a maximum number of steps to 500 and step length factor to 5.

3.6.2.2 Injections

After the discrete phase model is turned on, it is time to start injecting. On the 4 faces, 12 injections have been added, 3 on each face, at equal distance from each other at the height of 60mm. For all 12 injections, the diameter has been set to 0.5mm, 0.8mm and 0.11m and mass flow rate has been set to 16L/h, 21L/h and 26L/h with a temperature of 300K. The injection type is selected as Single because it provides the ability to put in the injections at specific locations according to project requirements. Additionally, a single injection type is used when we want to specify a single value for each initial condition. In my project, I have specified a mass flow rate which is used as an initial condition or value for all 12 injections, making a single type of injection a strong candidate for this project.

3.6.3 Materials

Materials options can be used to define the required materials. Various materials are available to choose from a material database. For my project, air and water vapour are selected as the material for the fluid domain, and the solid objects, like batteries, are made with copper. For the inert particle, it is water-air droplets with a density of 1000 kg/m3, and it will be injected through the nozzles that are defined in DPM.

3.6.4 Cell Zone Conditions and Boundary Conditions

The zones group in the Physics tab includes the cell zones, boundaries, and profiles menu. Using these three options, the user can do the following tasks.

- Cell zones allow the user to assign properties to cell zones.
- Boundaries assign boundary values to the surface of the model.
- Profile allows users to import data to specify cell zone and boundary conditions.

In our computational model, the cell zones menu allows users to assign materials to volumes or zones. When a cell zone is defined as a fluid, equations related to flow, energy, turbulence and so on are solved in the zone. If a cell is defined as a solid zone, Fluent only solves the energy equation in the zone.

Boundary Conditions is one of the critical components of any CFD simulations. Defining boundary conditions includes:

- Boundary values
- Boundary type
- Boundary location

These choices depend on various factors such as:

- Geometry
- Availability of data at boundary
- Numerical considerations

Moreover, boundary conditions are classified as external and internal. External boundaries are those where the mesh cells are attached to only one side of the boundary. These boundaries include various inlets, outlets, external walls and so on. At the inlets and outlet boundaries, we usually specify either the total pressure or the mass flow rate, or if the flow is incompressible, the mass flow rate can be replaced with velocity. In the project I am working on, there is no inlet, but an outlet is there, which is specified as a pressure outlet. All the other Faces and battery boundaries are considered walls. For the batteries which are named as B1, B2, B3, B4, B5, the heat flux values for them have been put 31W/m2 and 50W/m2, temperature value is 40C(313K) and material for these batteries have chosen as copper.

On the other hand, internal boundaries are those with cells attached to both sides of the boundary faces. The most used internal boundary types are the interior and wall. In my case, the interior boundary is defined as a Fluid domain with phases 1 and 2, where phase 1 is water vapour and phase 2 is air.

3.6.5 Solutions

3.6.5.1 Solution Methods

In the solution methods, the pressure-velocity coupling schemes are differentiated on the basis of how the pressure and velocity fields are updated when the pressure-based solver is used. They are either of the following types.

a) **Segregated type**

Pressure and velocity are updated substantially.

1- SIMPLE
2- SIMPLEC
3- PISO

b) Coupled

Pressure and velocity are updated simultaneously.

For solving the simulation for my project, I selected coupled because it is a default option and applicable to most of the simulation problems and coupled algorithm usually requires fewer iterations to converge. Moreover, from the spatial discretization options like gradient, pressure, momentum, volume fraction, turbulence kinetic energy, specific dissipation energy and energy, it is recommended to use default option, which is least squares cell based for the gradient, Presto for the pressure and compressive for the volume fraction and second order upwind for the energy equations. The reason for using second order upwind for the energy equations is for obtaining the most accurate solution on a suitable refined numerical mesh.

3.6.5.2 Solution Initialization

Ansys Fluent solves governing equations iteratively to calculate the solution. However, before starting any simulation, Fluent requires the solution variables to be initialized before starting any CFD simulation. Initialization primarily means that each individual cell must be assigned a value for every solution variable to serve as an initial guess for the solution. The general parameters considered for the initialization are:

1. Pressure
2. Velocity
3. Temperature (if energy equation is ON)
4. Turbulence parameters (based on the turbulence model used)

3.6.5.2.1 Types of Initializations

1. Standard initialization
2. Hybrid initialization

For my project, I am using hybrid initialization because it provides a closer initial guess to the solution and in Hybrid initialization, The initial solution is obtained by solving a

Laplace equation under suitable boundary conditions. Getting an adequate starting solution (or one that is at least superior to the standard initialization) is the goal here.

3.6.5.3 Calculation Activities

Calculation activities are the last step in the solution method task. All the other options under the calculation activities have been set to default apart from the run calculation in which the number of time steps was set to 100, time step size to 0.01, and max iterations/time setup to 5. Below is the way to calculate the number of time steps.

In the injection tab, I have selected a start and stop to 0 and 1 second. So, the number of time steps=$\frac{Stop\ time - start\ time}{time\ step\ size}$=$\frac{1-0}{0.01}$=100

Furthermore, for the time-step size, the lesser the value is, the finer the simulation results, which means putting 0.1 for the time-step size means less accuracy in simulation results as compared to 0.01. However, the decrease in time-step size means if I put 0.01 with 0 and 1-second start and stop time, it will increase the number of times steps and the simulation time.

Lastly, for the max iteration/time setup, putting a number 5 with 100-time steps means the total max iteration will be 500. Below is the way to total max iterations.

Total max iteration=maximum iteration per time step x number of time steps

Total max iteration= 5 iterations/time steps x 100-time steps

Total max iteration= 500 iterations.

4. Results and Discussion

4.1 CFD Post Processing

Post-processing is a general term that includes all the way we interact and examine simulation results. Post-processing can be broadly classified into two categories:

- Qualitative post-processing
- Quantitative post-processing

Qualitative post-processing involves contours, vectors, pathlines. etc., which can be used to visualise what the flow looks like, and it is primarily used to gain a visual inside into complex flow phenomena. On the other hand, Quantitative post-processing involves XY plots, histograms, average reports, forces, fluxes, etc., which can be used to get numerical data to identify trends, patterns and comparison with analytical experiment or similar simulation results. Post-processing is a general term that includes all the ways we interact and examine simulation results.

4.1.1 Temperature Contours and Vector Plots at Heat Flux=31W/m2

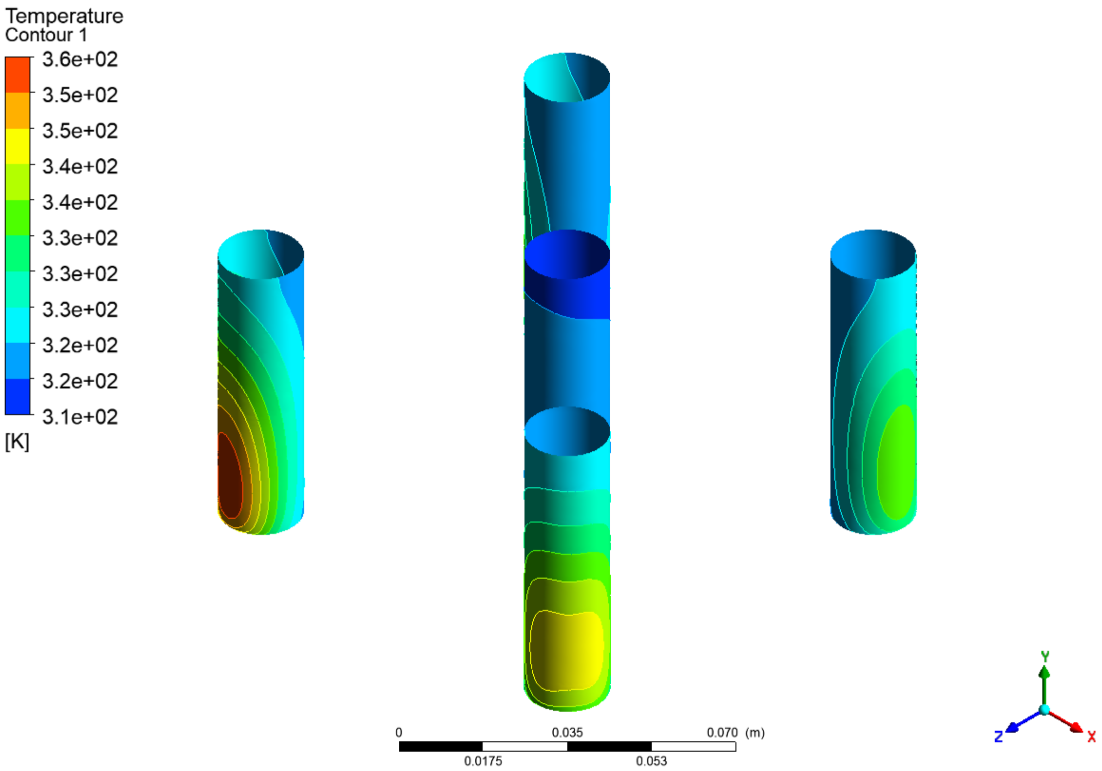


Figure 10 Batteries Contour at Mass Flow Rate=16L/h

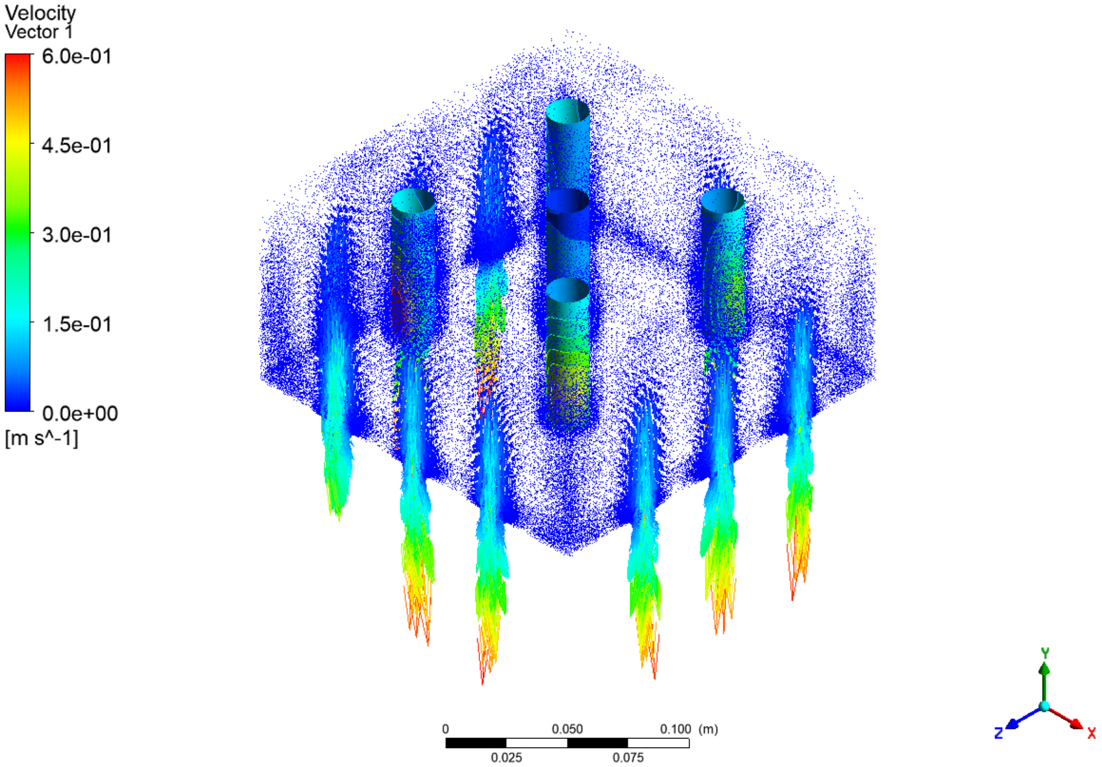


Figure 11 Vector Plot at Mass Flow Rate=16L/h

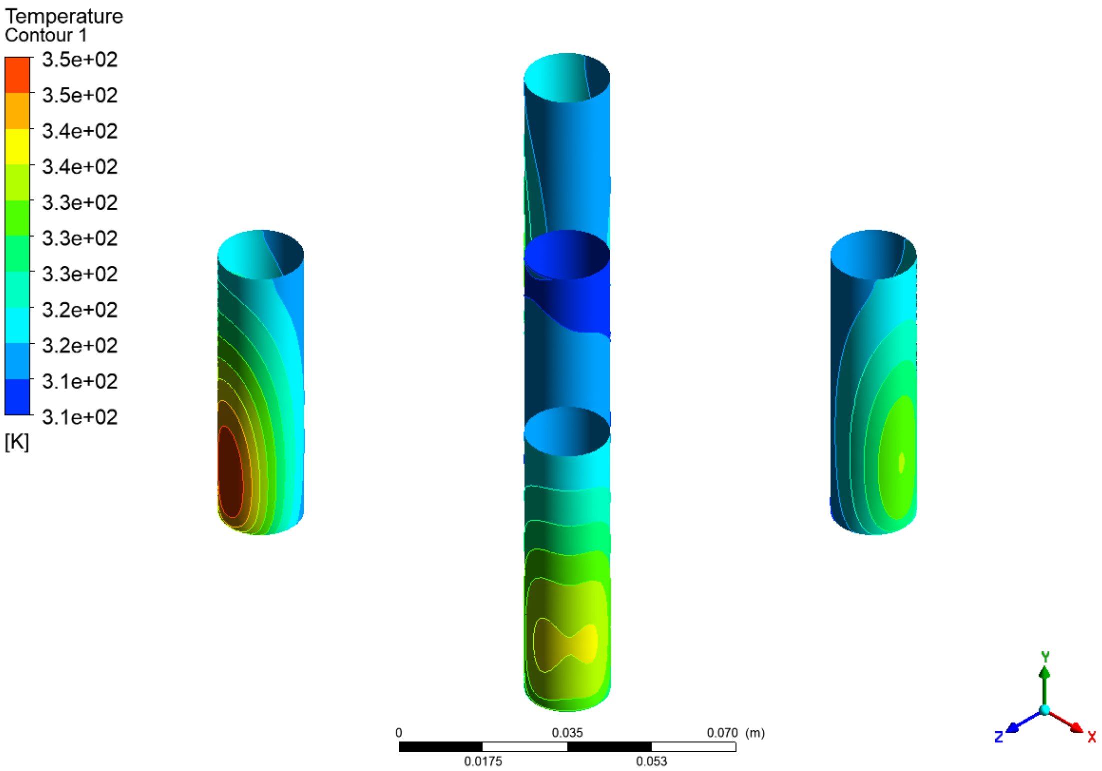


Figure 12 Batteries Contour at Mass Flow Rate=21L/h

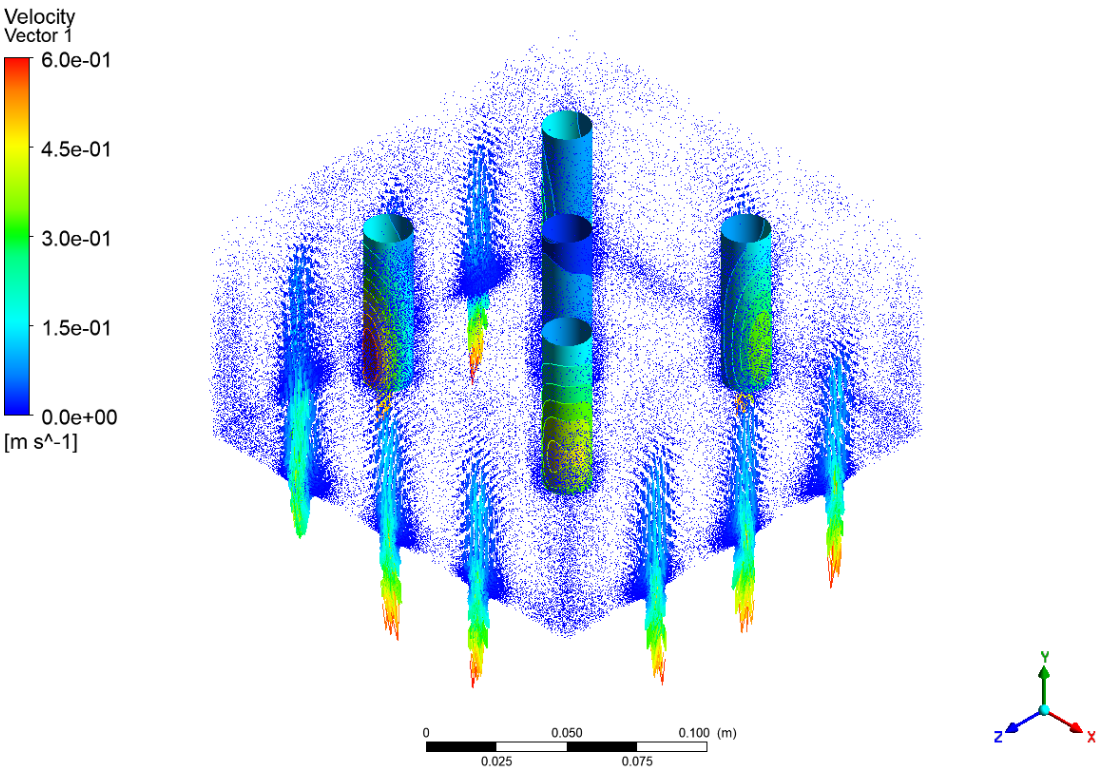


Figure 13 Vector Plot at Mass Flow Rate=21L/h

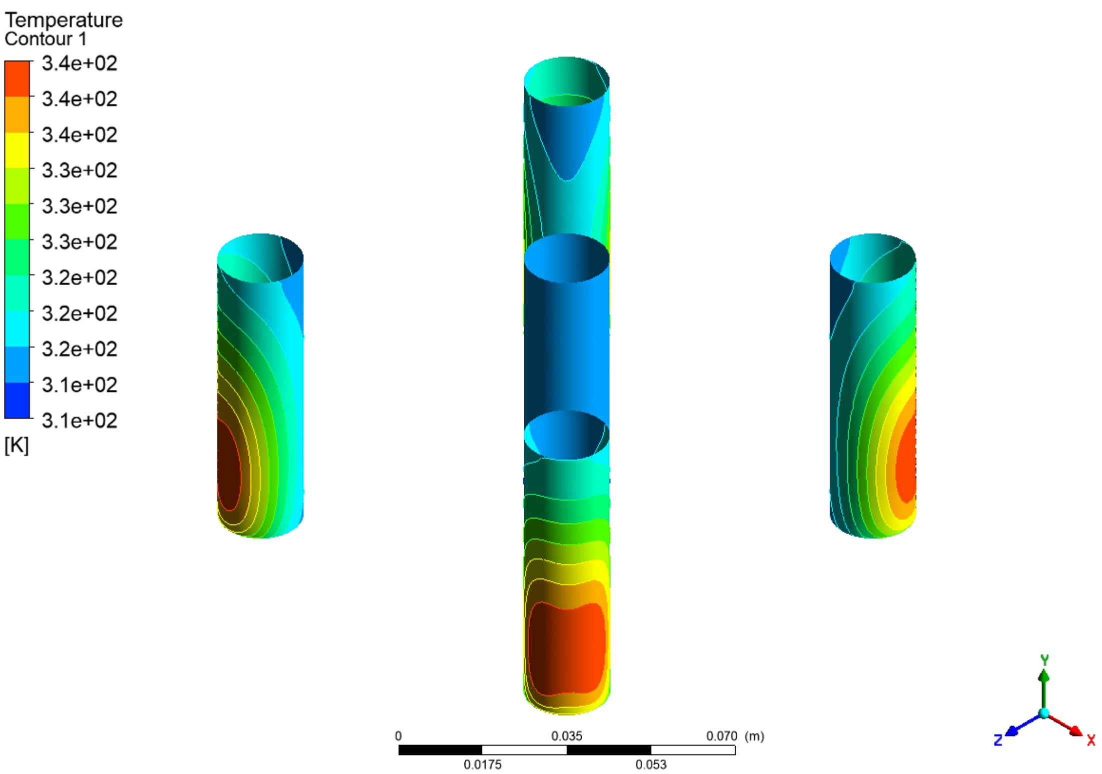


Figure 14 Batteries Contour at Mass Flow Rate=26L/h

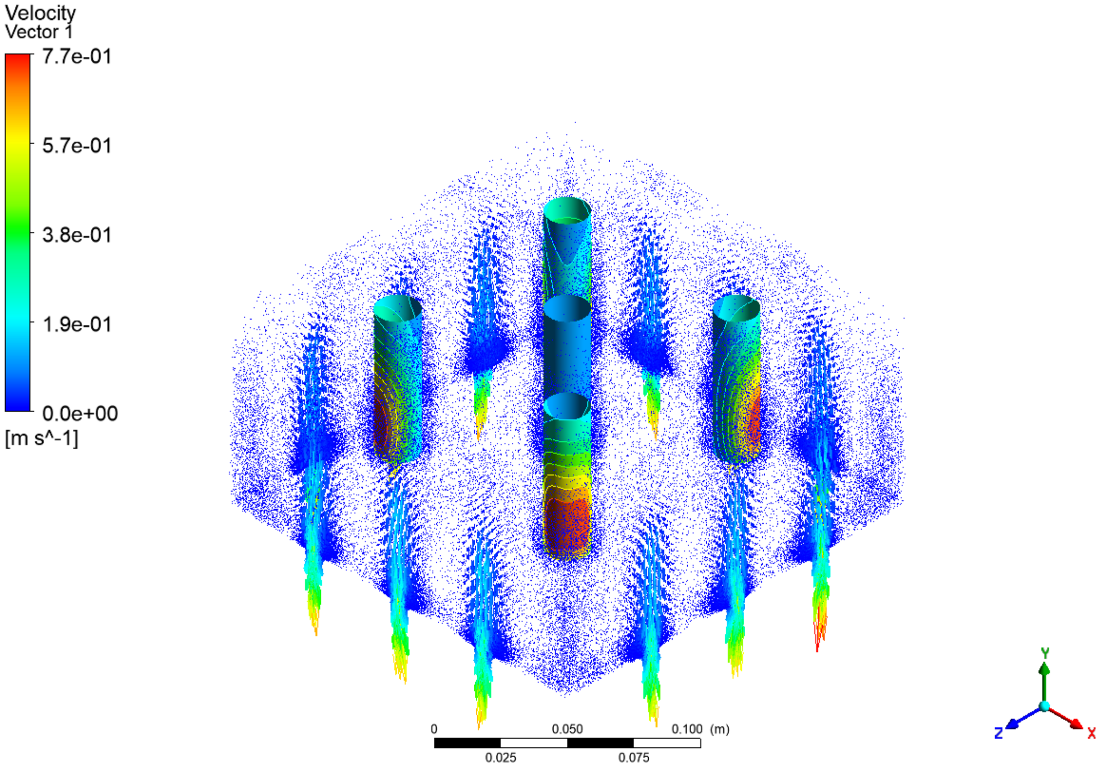
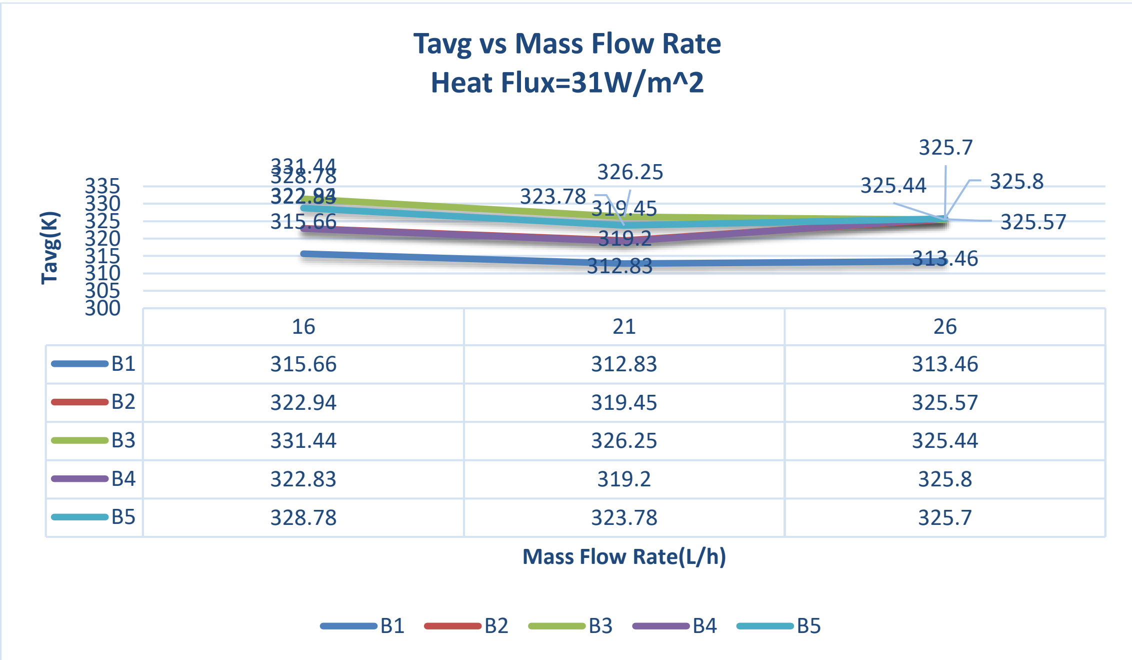


Figure 15 Vector Plot at Mass Flow Rate=26L/h



	16	21	26
B1	315.66	312.83	313.46
B2	322.94	319.45	325.57
B3	331.44	326.25	325.44
B4	322.83	319.2	325.8
B5	328.78	323.78	325.7

Figure 16 Tavg vs Mass Flow Rate at Heat Flux=31W/m2

Figures 10 to 15 show the contour and vector plots when the mass flow rate changes from 16L/h to 21L/h and then 21L/h to 26L/h by keeping the heat flux constant at 31W/m2. This time, a line graph, Figure 16, shows that batteries' temperature at a mass flow rate of 21L/h is less for all batteries than at a mass flow rate of 16L/h and 26L/h. This is likely due to different reasons, which are discussed below.

- The flow regime within the cooling system may change due to a change in mass flow rate. Different flow regimes, like laminar and turbulent flow, have various properties that affect heat transfer. The flow regime might be less effective at removing heat from the batteries at the lower and higher flow rates (16 L/h and 26 L/h), increasing temperature. A more advantageous flow regime might be present at 21 L/h, leading to better heat transfer and lower temperatures. 21L/h and 31W/m2 were the numbers given at the start of the simulation, and it is evident from the line graph Figure 16 that these numbers suit best for this simulation and also help to get a lower temperature across the surfaces of the batteries.
- Another reason the temperatures of the batteries can be impacted is uneven flow distribution. Flow distribution is less efficient at 16 L/h and 26 L/h, heating some batteries. The flow pattern may be more uniform at 21 L/h, resulting in lower temperatures. This aspect has also been proved in other studies. For instance, Ismail et al. (2013) found that temperature variations inside batteries can result in uneven temperature distribution, resulting in inconsistent charge/discharge behaviour throughout the pack. In the worst-case scenario, thermal runaway may develop in a cell when temperatures are not effectively controlled, potentially resulting in fire and explosion. As a result, battery heat management is critical for hybrid and electric cars to maintain optimal performance (Ismail et al., 2013).

4.1.2 Temperature Contours and Vector Plots at Heat Flux=50 W/m2

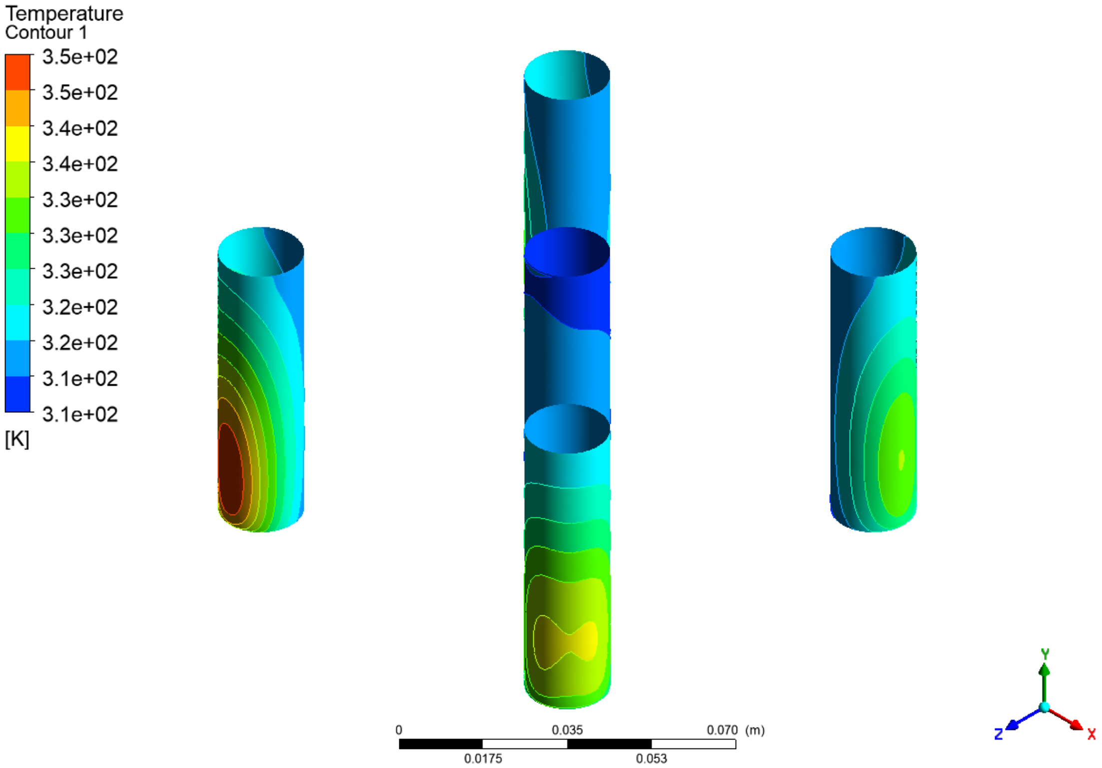


Figure 17 Batteries Contour at Mass Flow Rate=16L/h

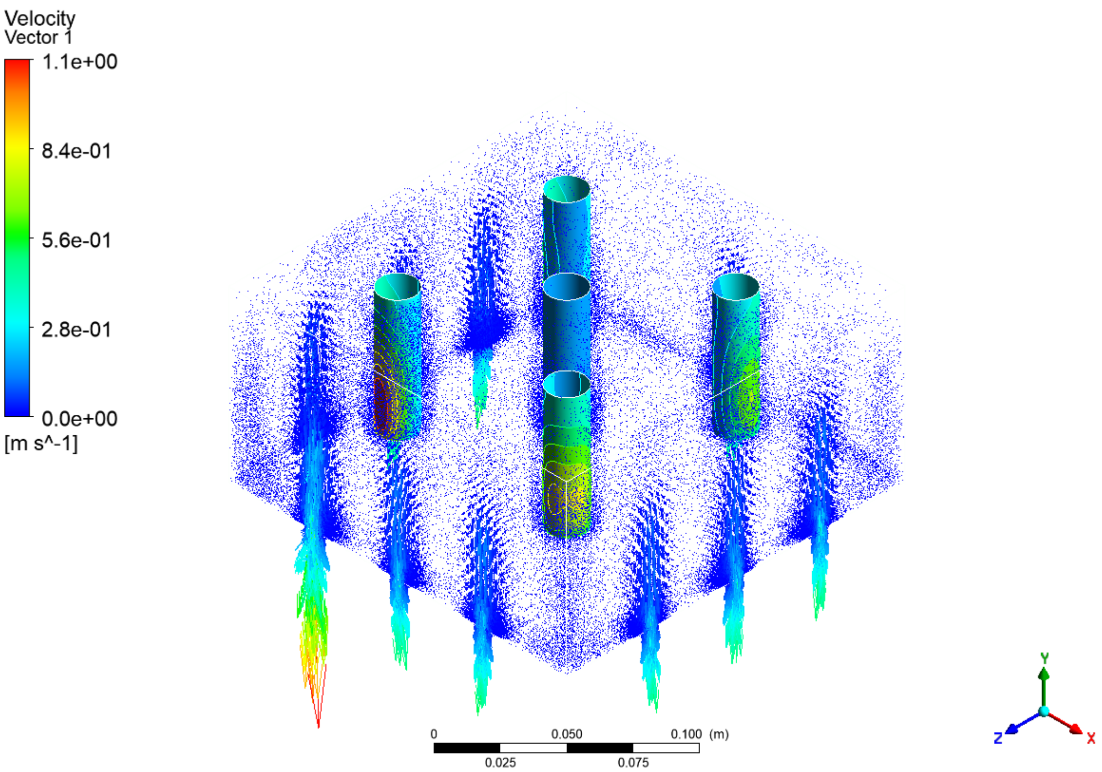


Figure 18 Vector Plot at Mass Flow Rate=16L/h

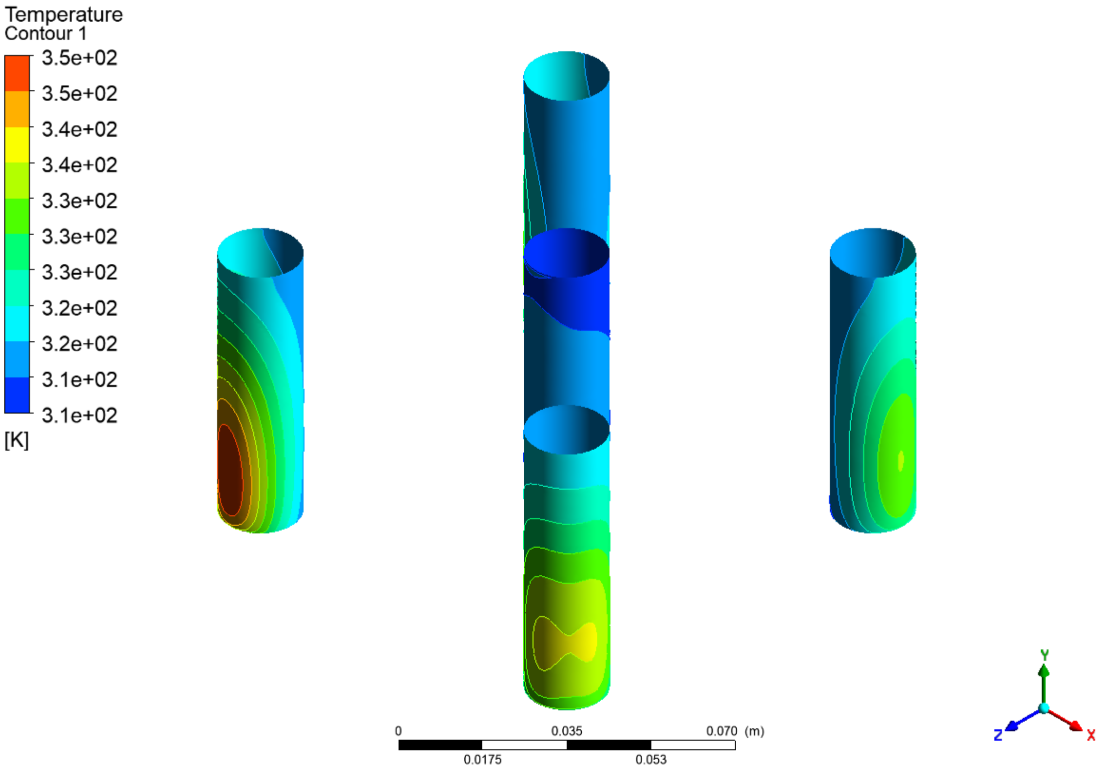


Figure 19 Batteries Contour at Mass Flow Rate=21L/h

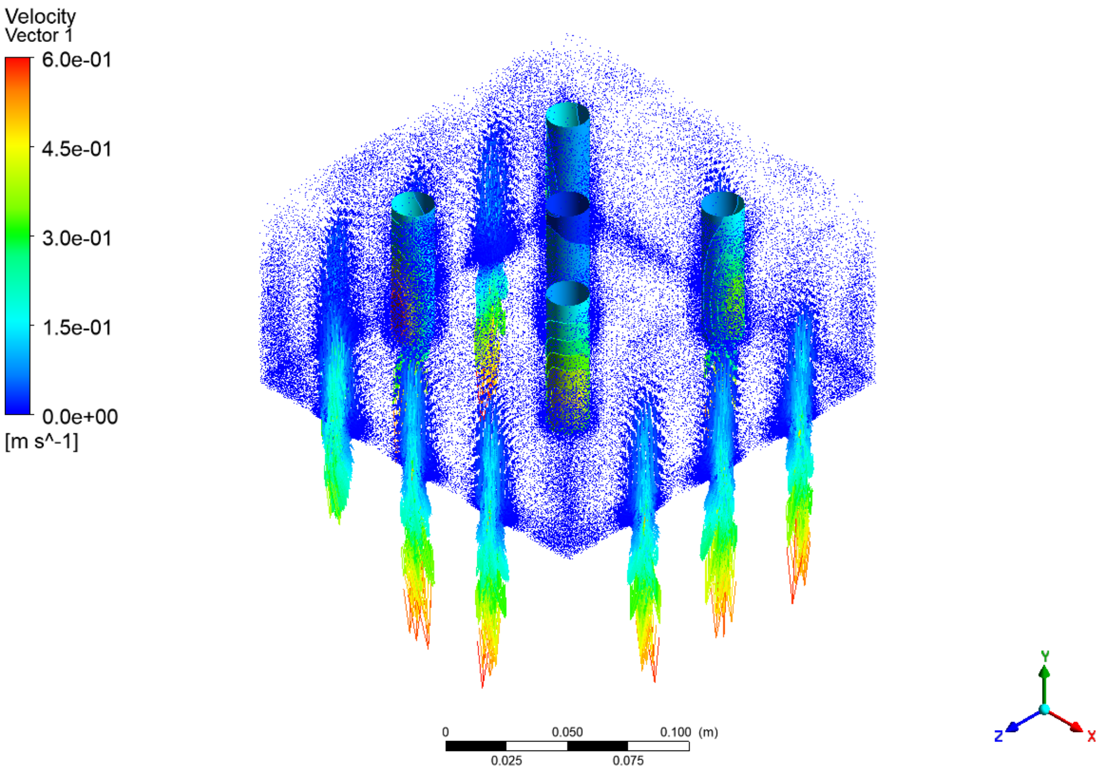


Figure 20 Vector Plot at Mass Flow Rate=21L/h

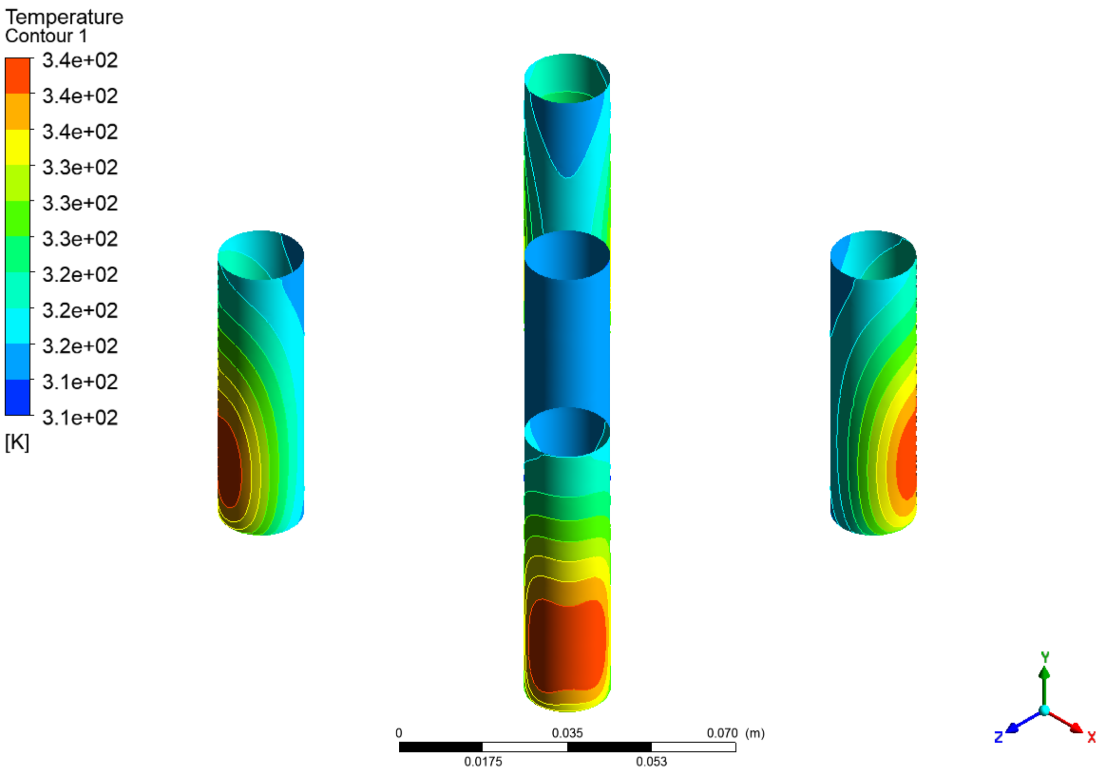


Figure 21 Batteries Contour at Mass Flow Rate=26L/h

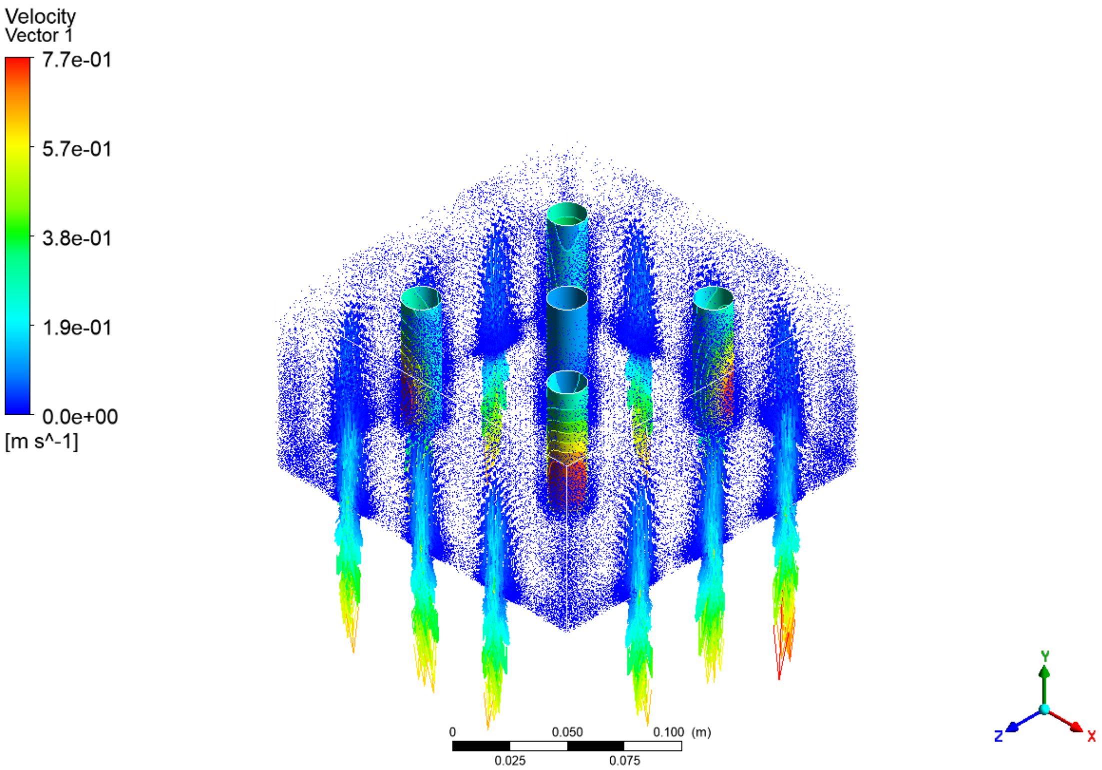
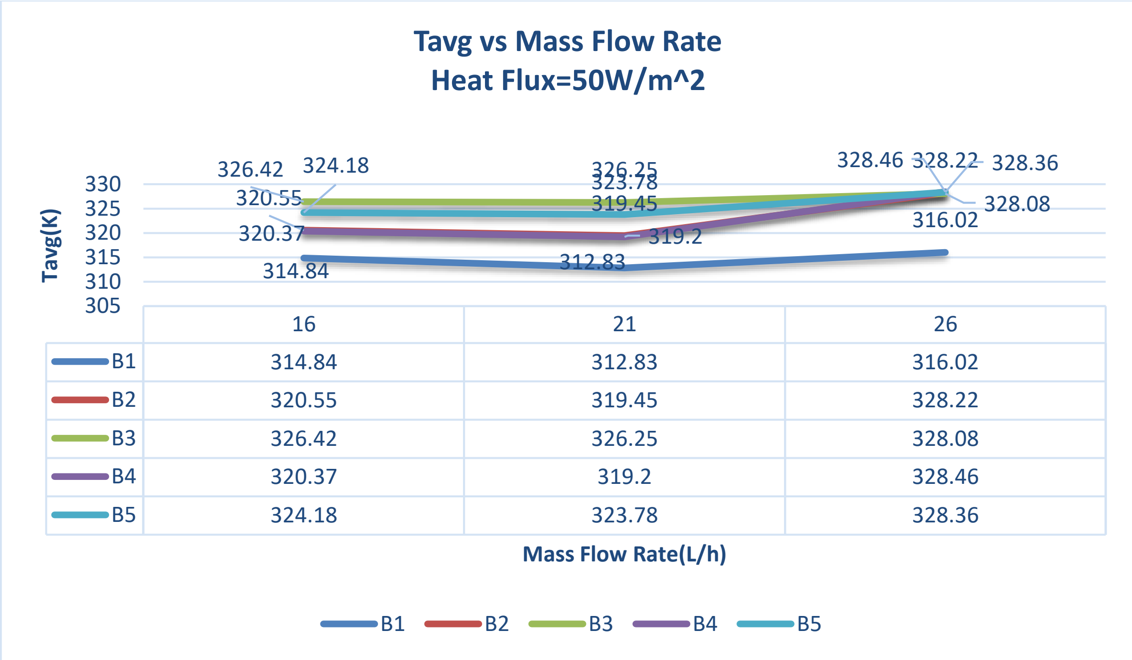


Figure 22 Vector Plot at Mass Flow Rate=26L/h



	16	21	26
B1	314.84	312.83	316.02
B2	320.55	319.45	328.22
B3	326.42	326.25	328.08
B4	320.37	319.2	328.46
B5	324.18	323.78	328.36

Figure 23 Tavg vs Mass Flow Rate at Heat Flux=50W/m2

Figures 17 to 22 show the contour and vector plots when the mass flow rate changes from 16L/h to 21L/h and then 21L/h to 26L/h by keeping the heat flux constant at 50W/m2. This time, in a line graph, Figure 23, shows that batteries' temperature at a mass flow rate of 21L/h is less for all batteries than at a mass flow rate of 16L/h and 26L/h. This is also due to the flow regime, flow distribution, and heating resistance I have discussed above for a mass flow rate of 21L/h and heat flux at 31W/m2. Regarding the flow regime, a more significant mass flow rate often results in better cooling performance, lowering the battery's temperature. This is because more significant cooling fluid can carry away more heat than the battery produces (Mukone et al., 2021). An extremely high flow rate, on the other hand, may cause other problems, such as a more significant pressure drop and disturbance.

On the other hand, an inconsistent flow distribution in the battery might cause hot spots if particular parts are not effectively cooled. This can raise the battery's temperature and cause thermal runaway (Mukone et al., 2021). As a result, a good flow dispersion is critical for keeping a consistent temperature throughout the battery. A more considerable internal resistance equals more heat generation, which raises the battery's temperature. As a result, lowering the resistance can aid in cooling the battery. However, the batteries' temperature at a mass flow rate of 16L/h and 26L/h shows an increase in temperature as evident in Figure 23.

4.1.3 Temperature Contours and Vector Plots at diameter=0.5mm

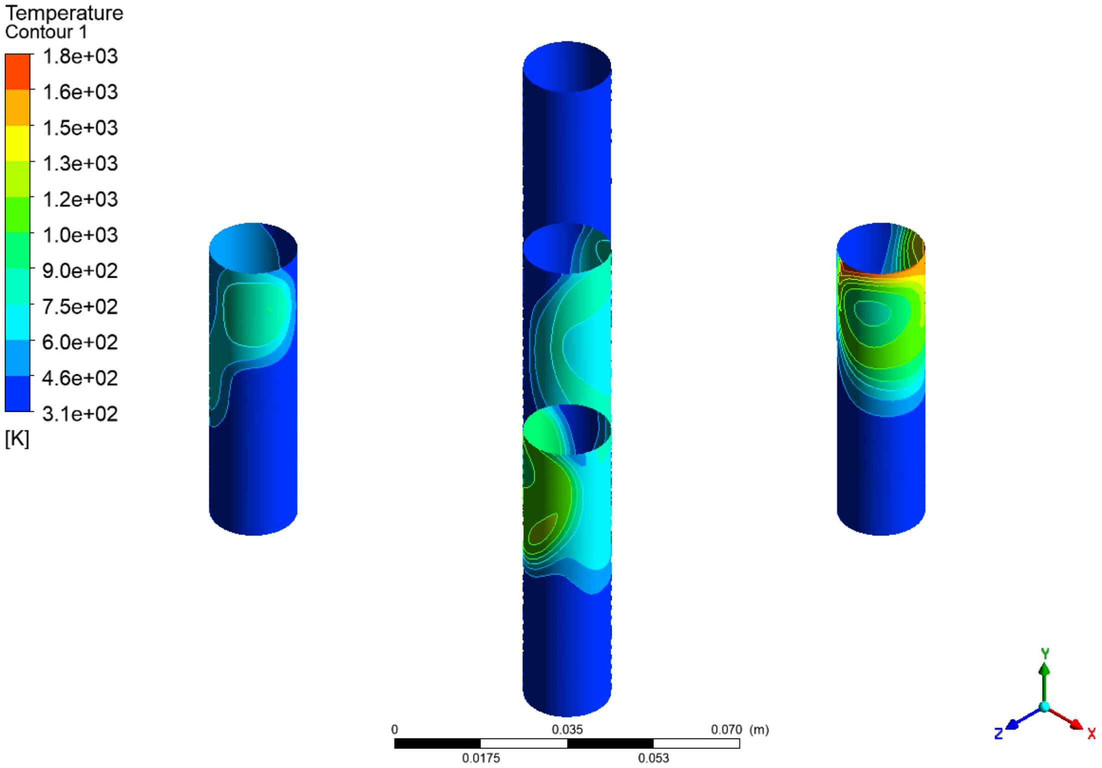


Figure 24 Batteries Contour at D=0.5mm

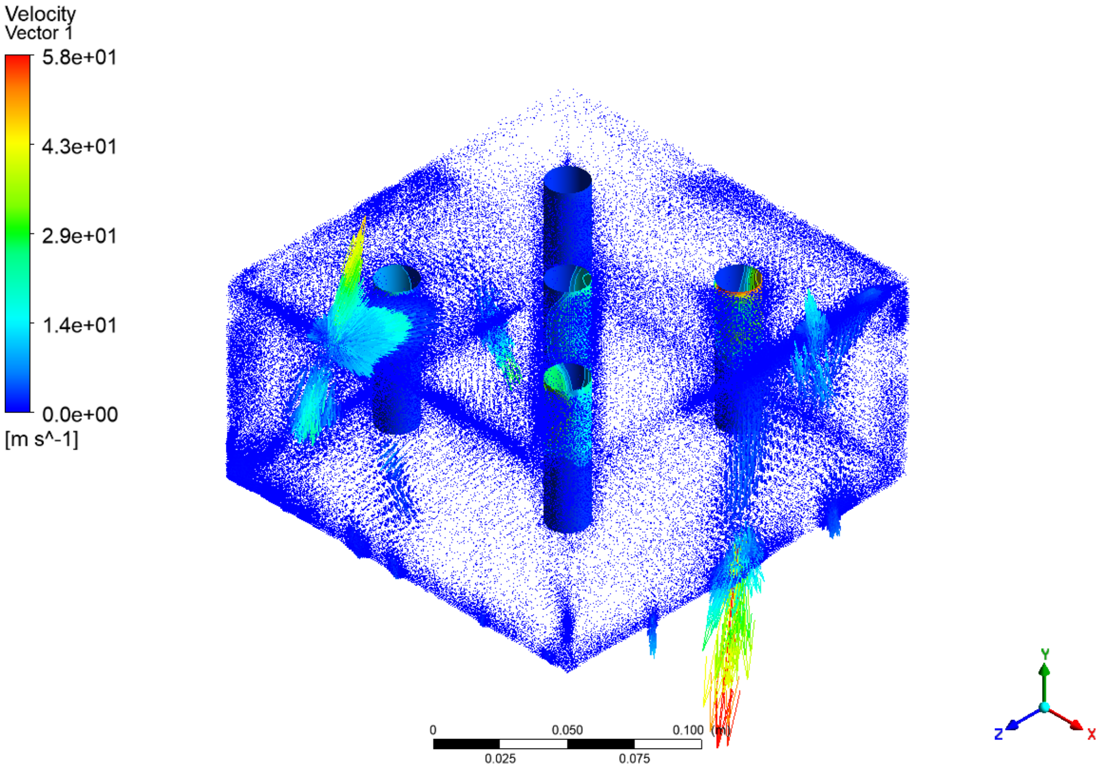


Figure 25 Vector Plot at D=0.5mm

4.1.4 Temperature Contours and Vector Plots at diameter=0.8mm

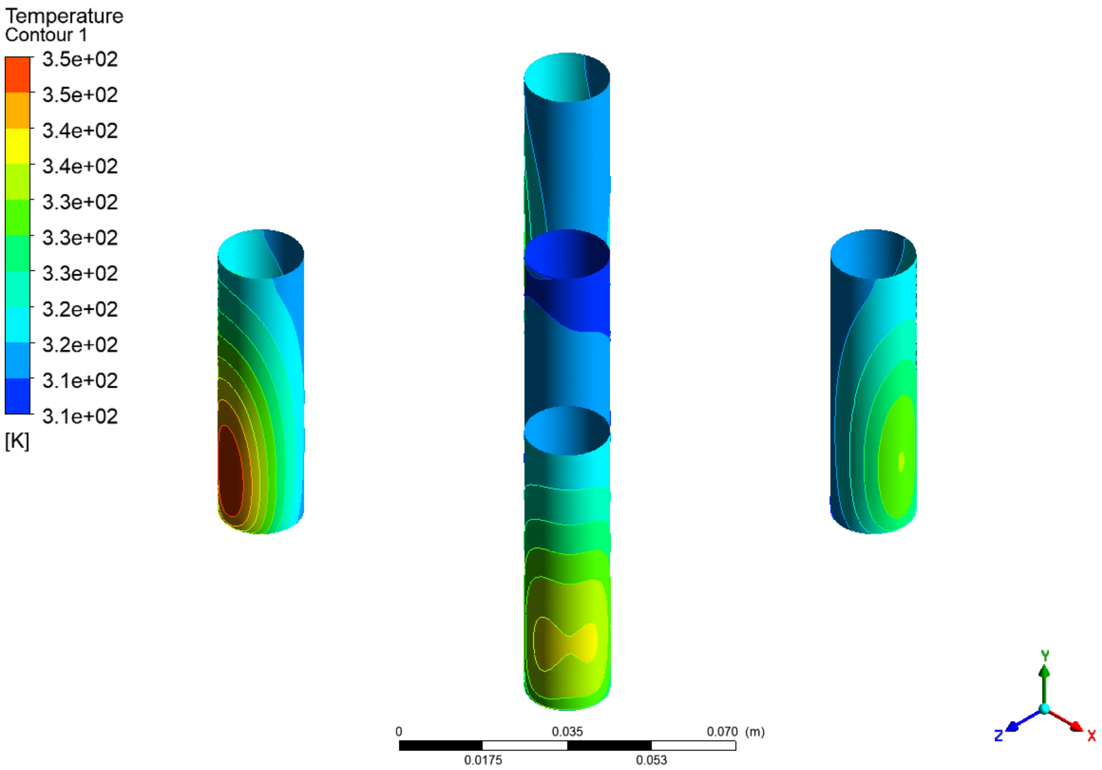


Figure 26 Batteries Contour at D=0.8mm

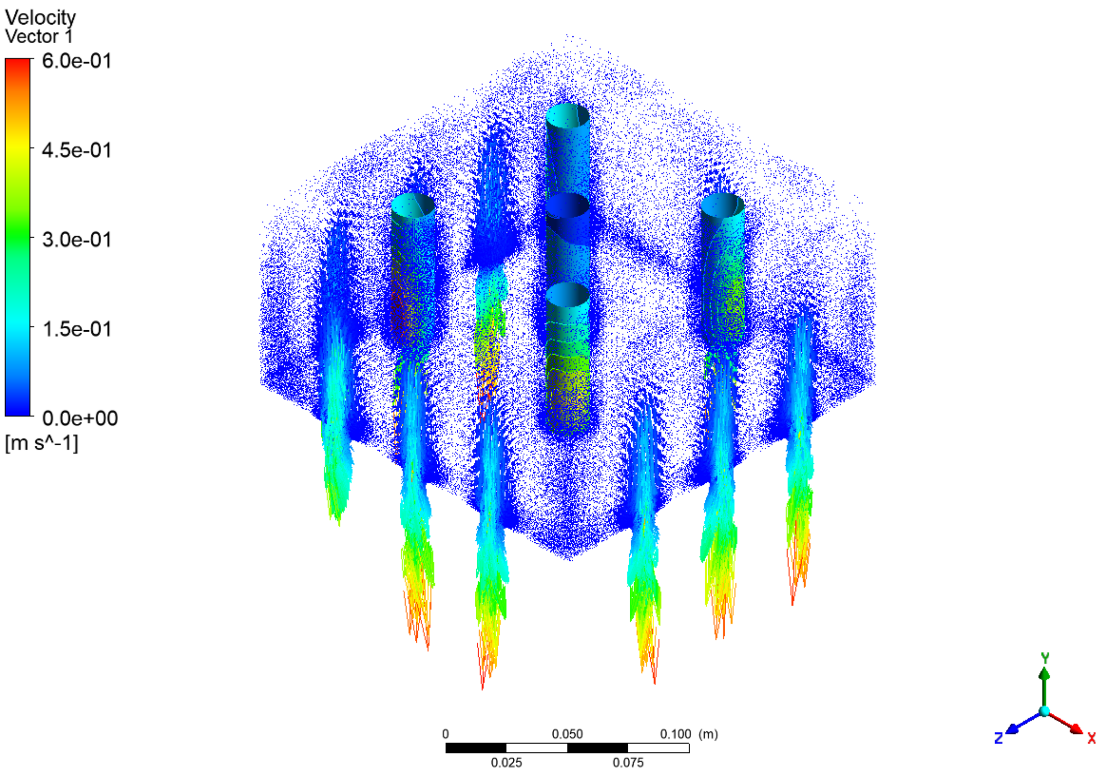


Figure 27 Vector Plot at D=0.8mm

4.1.5 Temperature Contours and Vector Plots at diameter=0.11mm

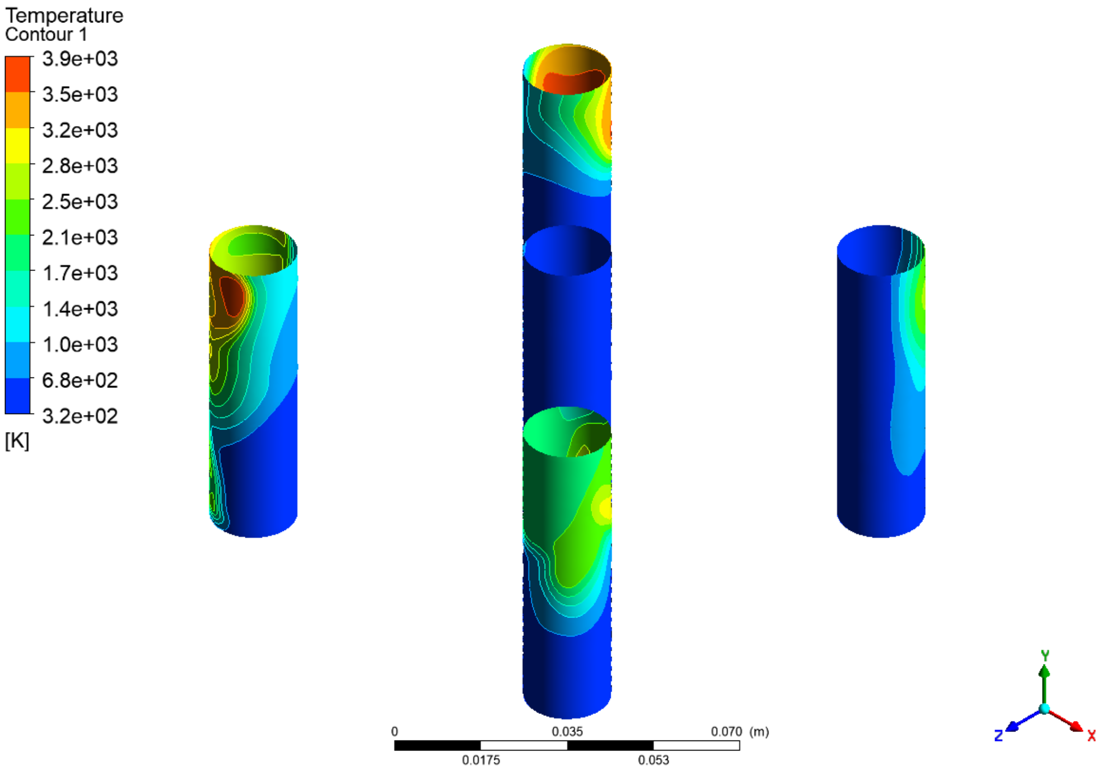


Figure 28 Batteries Contour at D=0.11mm

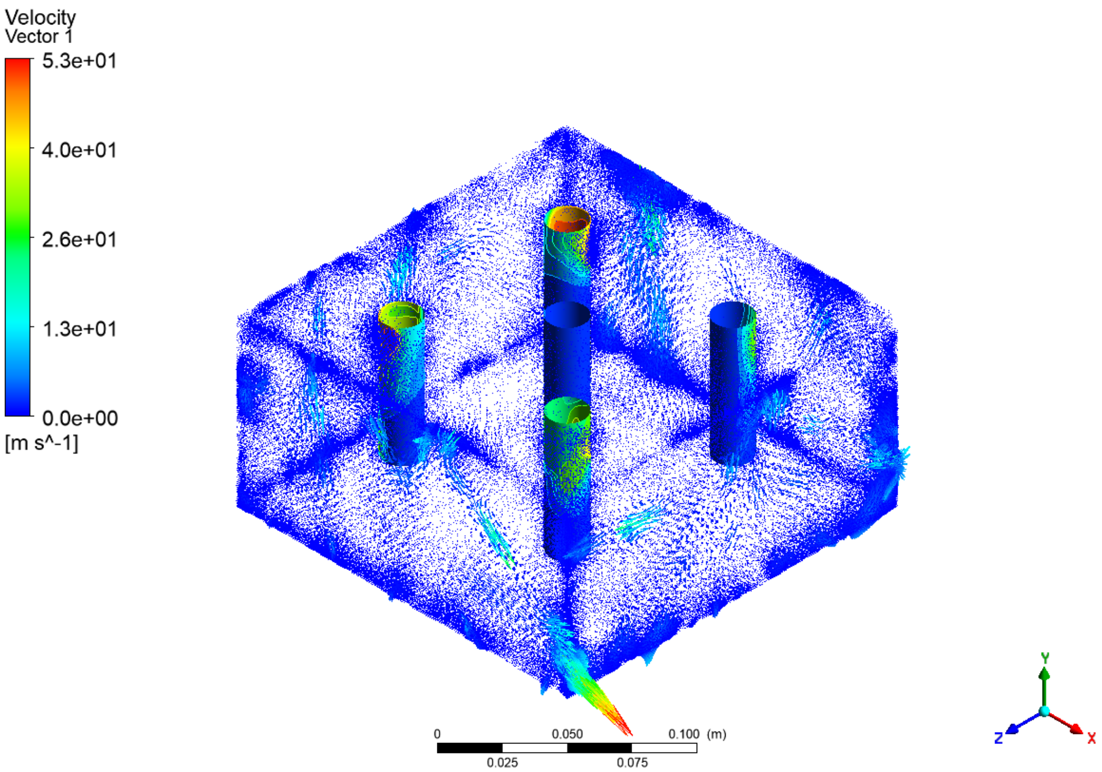


Figure 29 Vector Plot at D=0.11mm

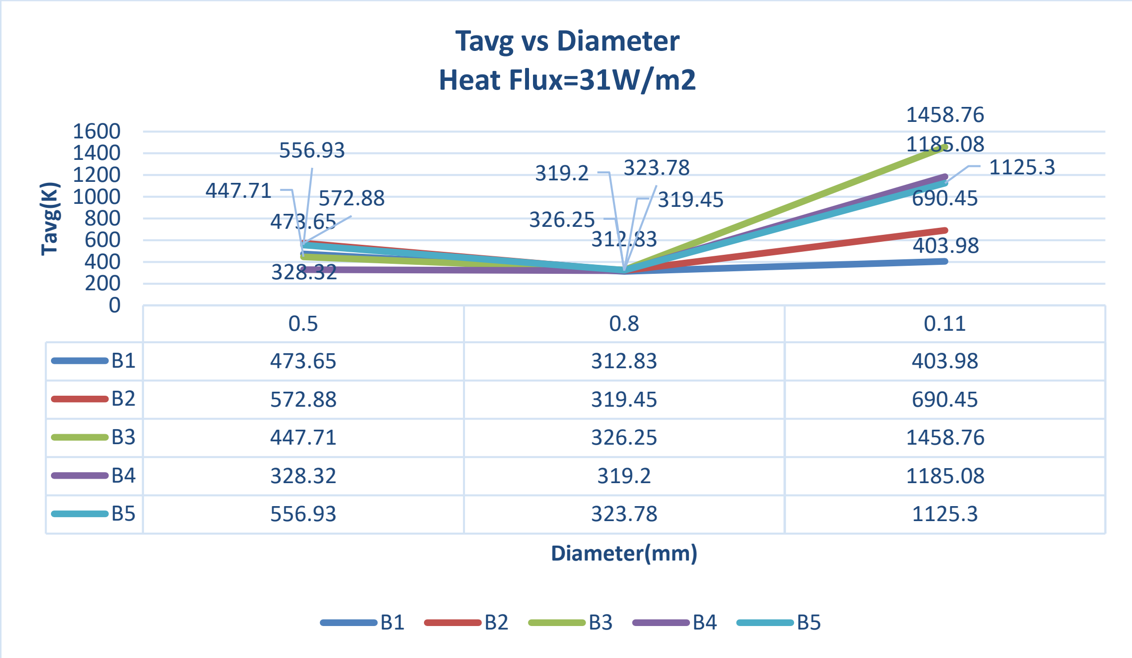


Figure 30 Tav vs Diameter at Heat Flux=31W/m2

As shown in Figure 30, the battery temperatures at the Injection Particle with a nozzle diameter of 0.5mm were high. However, the battery's temperature reduced when the nozzle diameter was 0.8mm but rose again when the diameter was 0.11 mm. This indicated that the appropriate mass flow rate and heat flux are responsible for decreasing the temperature distribution across the surfaces of the batteries and that an appropriate nozzle size is also required, which in this case is 0.8mm, as depicted in Figure 30. Furthermore, when batteries are charged or drained, they generate heat owing to internal resistance. The design of the battery, the pace of charge or release, and the effectiveness of the battery's cooling system all significantly impact heat production (Kumar et al., 2018). However, this behaviour of the battery can be viewed from a different perspective. For instance, increasing the nozzle hole size can result in more pressure being injected, affecting the battery's performance (Kumar et al., 2018). This can be used to explain why the temperatures of the battery decreased when the diameter of the inject particles nozzle was increased from 0.5mm to 0.8mm. However, explaining why the temperatures increased when the diameter increased to 0.11mm can be difficult when considering the diameter of the nozzle. In this regard, the main argument may be based on the battery's internal heat resistance capabilities. This may also include the design of the battery and the technology used in the battery.

4.1.6 Temperature Contours and Vector Plots at Mass Flow Rate=21L/h

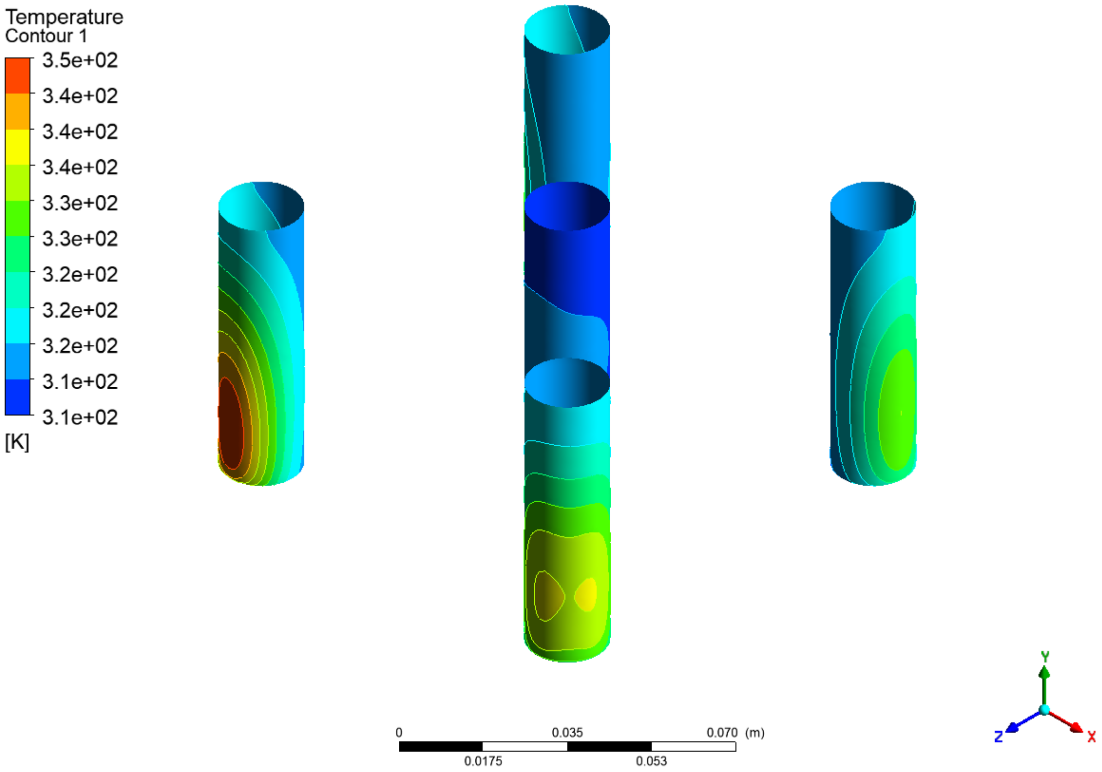


Figure 31 Batteries Contour at Heat Flux=20W/m2

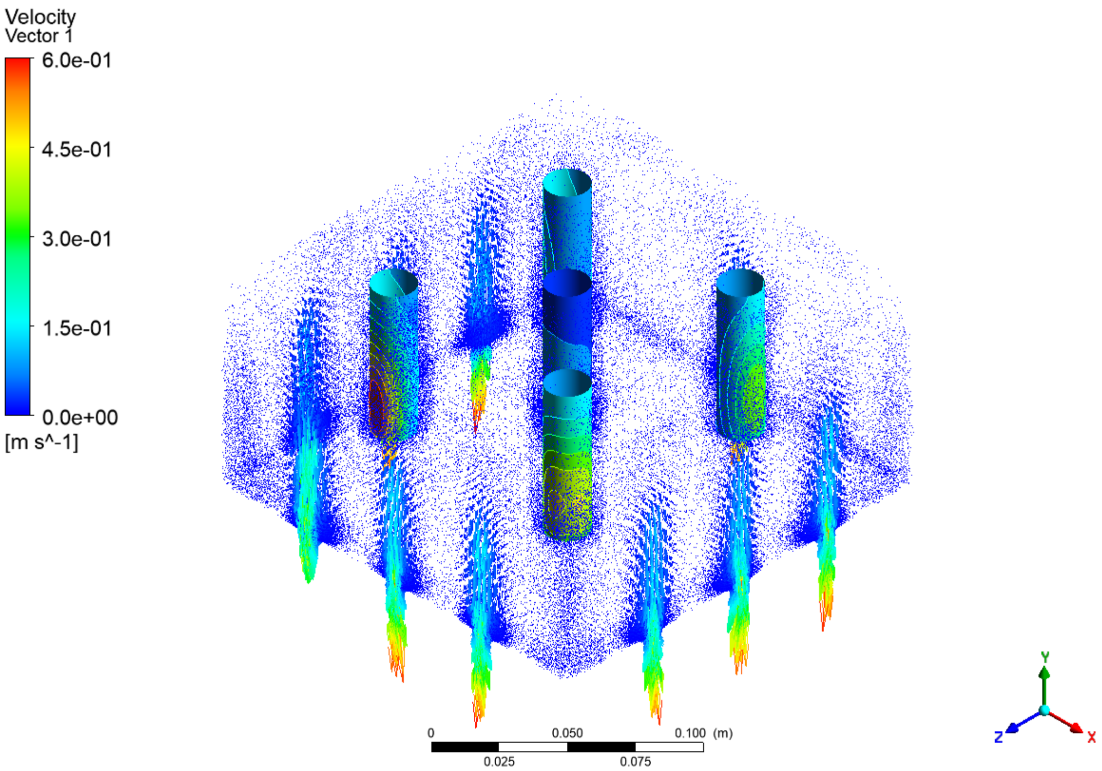


Figure 32 Vector Plot at Heat Flux=20W/m2

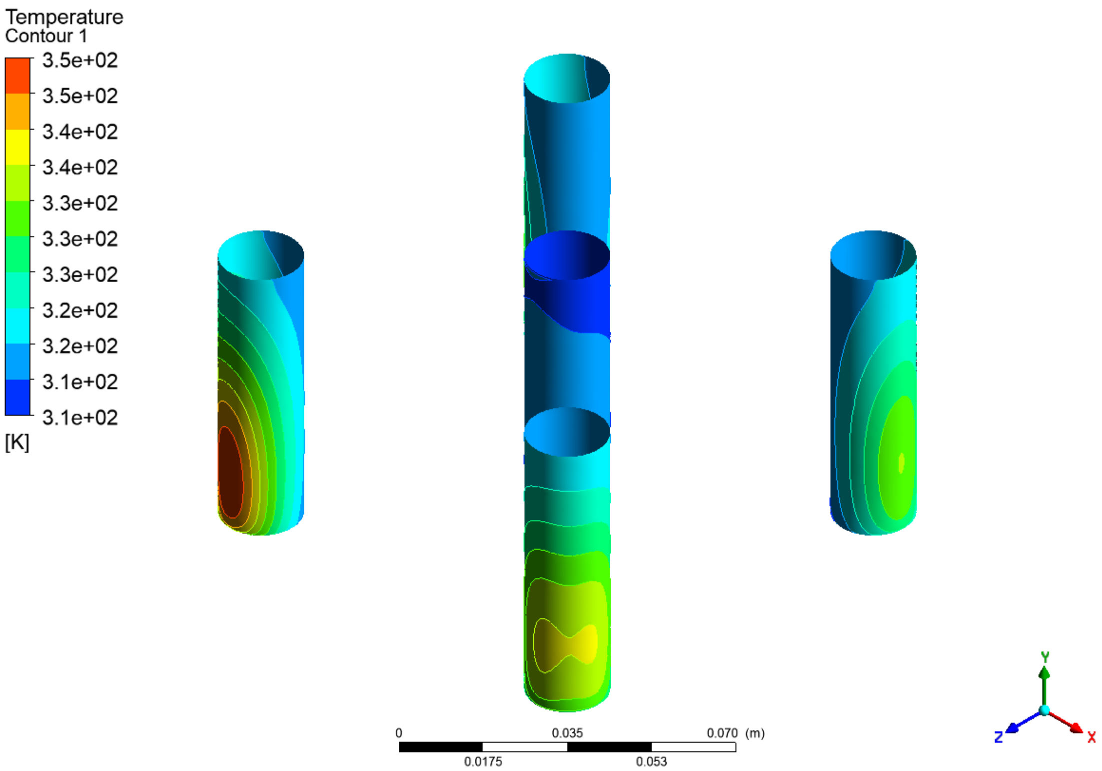


Figure 33 Batteries Contour at Heat Flux=31W/m2

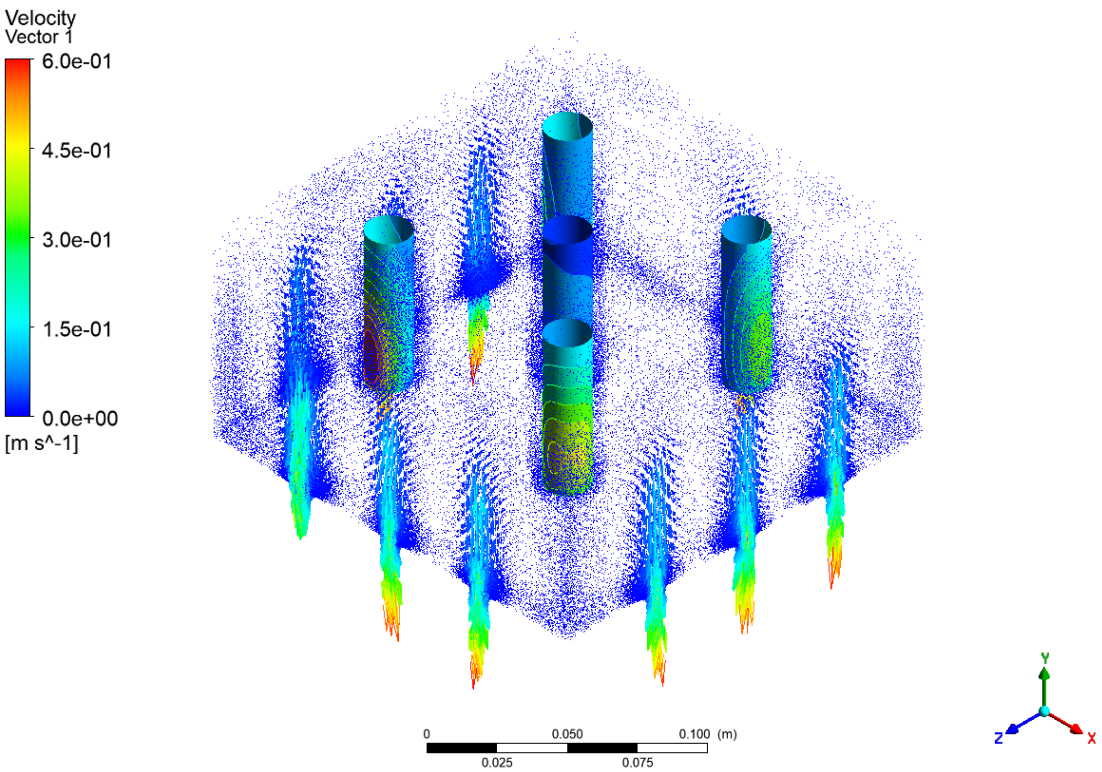


Figure 34 Vector Plot at Heat Flux=31W/m2

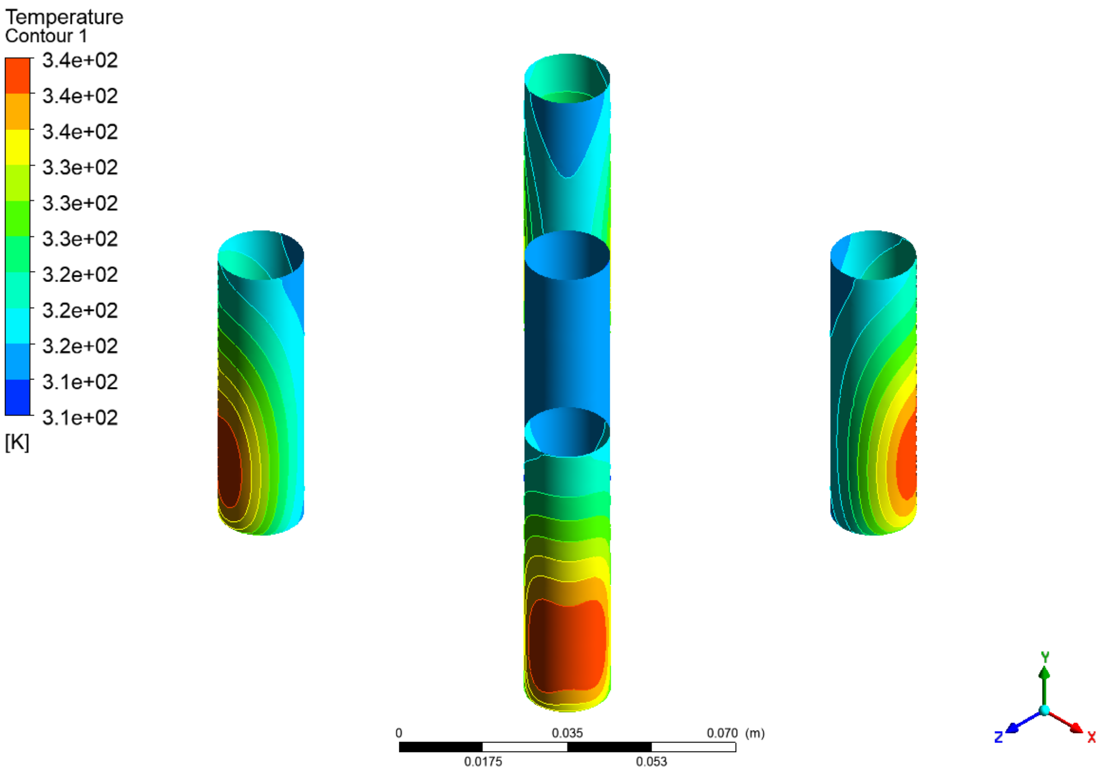


Figure 35 Batteries Contour at Heat Flux=50W/m2

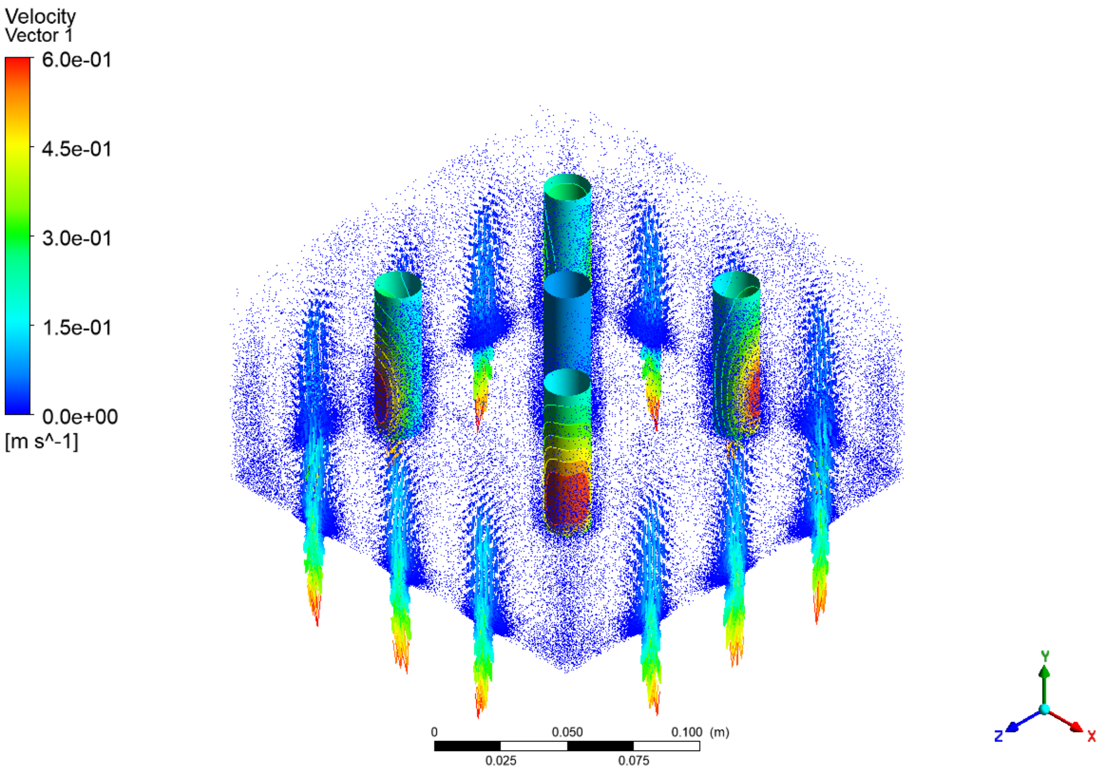
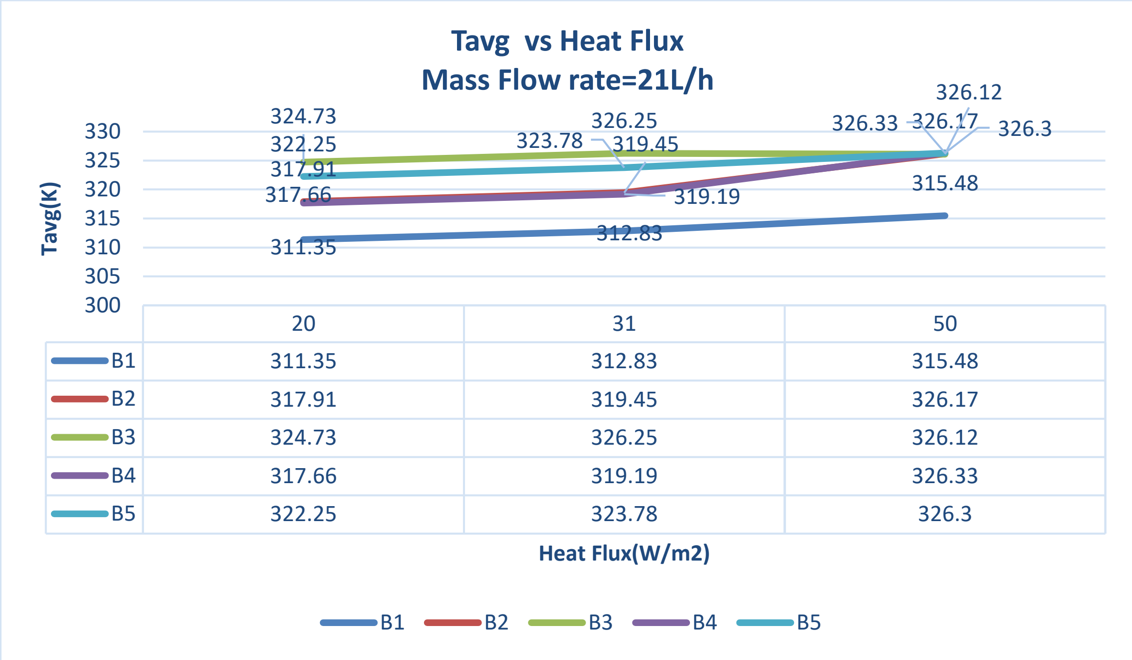


Figure 36 Vector Plot at Heat Flux=50W/m2



	20	31	50
B1	311.35	312.83	315.48
B2	317.91	319.45	326.17
B3	324.73	326.25	326.12
B4	317.66	319.19	326.33
B5	322.25	323.78	326.3

Figure 37 Tavg vs Heat Flux at Mass Flow Rate=21L/h

Figures 31 to 36 show the batteries' contour and vector plots when the mass flow rate is constant at 21L/h and heat flux changes from 20W/m2 to 31W/m2 and then 31W/m2 to 50W/m2. This indicates removing heat from the batteries by adding a constant volume of cooling fluid (Water vapour and air). The volume of fluid moving through the system per unit of time is represented by the mass flow rate, which, in this instance, is constant. Applying more heat per unit area to the surface of the batteries by raising the heat flux from 20W/m2 to 31W/m2 and then 31W/m2 to 50W/m2 will increase the batteries' temperature, as depicted in Figure 37. Heat naturally moves from a region of greater temperature to a lower temperature by the laws of thermodynamics. Heat is transmitted from the batteries to the cooling fluid when it comes to battery cooling. More heat is produced inside the batteries and delivered to the cooling fluid with a constant mass flow rate and an enhanced heat flux. However, while the cooling capacity is constant, the cooling system may need to eliminate this extra heat appropriately.

5. Conclusion

Various results have been obtained in this study. For example, when examining the contour and vector plots, the results indicate that a constant heat flux will probably result in a constant specific heat capacity. Increasing the heat mass, however, is likely to be distributed over, resulting in a temperature decrease. The results have, however, also shown that the cooling regime in the battery's cooling system is likely to change due to the mass flow rate. Therefore, while planning and controlling heat flow within a battery, the mass's flow rate is also crucial. The findings have also shown that uneven flow distribution of heat in the battery will likely affect the temperature of the battery. The vector plots and temperature contours at different heat fluxes provided proof of this. The results show that a higher mass flow rate will probably produce a cooling effect that benefits the battery. However, heat spots will likely impact the battery's ability to cool. Lastly, the results were also utilised to investigate the possibility that the nozzle's width may impact the battery's ability to produce heat. Most significantly, these findings have proved practical in designing battery systems that can be effective today. This is mainly because batteries have slowly found their way into the most essential devices, which include vehicles. Management of heat, therefore, proves to be very important for batteries.

Computational models can also be used to optimise the design of rechargeable batteries and assess the distribution of temperatures and heat generation to size any cooling arrangement under different operating situations. The establishment of X-ray diffraction analysis for studying cell materials is one promising field for future research. This approach can provide valuable insights into the structure of crystals and the composition of substances, enabling improved characterisation and model accuracy. Another potential field for future advancement is the model's capacity to simulate behaviour at greater C-rates. This could include fine-tuning the model's parameters, integrating more comprehensive heat production and transfer models, or considering additional electrochemical processes or physical occurrences to better record the battery's actions under these conditions.

References

Abderrahim, S. (2021). *Should I use Pressure Based Solver or a Density-Based Solver?* ResearchGate Find and share research. https://www.researchgate.net/post/Should_I_use_Pressure_Based_Solver_or_a_Density-Based_Solver

AKG of America. (2018). *AKG leads electric vehicle battery cooling research*. OEM Off-Highway. https://www.oemoffhighway.com/electronics/power-systems/batteriescapacitors/press-release/21029995/akg-of-america-akg-leads-electric-vehiclebattery-cooling-research

Ansys Innovation Space. (2023). *Mesh Generation Using ANSYS Fluent Water Flow*. courses.ansys.com. https://courses.ansys.com/index.php/courses/mesh-generationusing-ansys-fluent-watertight-workflow/

Bamrah, P., Kumar Chauhan, M., & Singh Sikarwar, B. (2022). CFD analysis of battery thermal management system. *Journal of Physics: Conference Series*, *2178*(1), 012035. https://doi.org/10.1088/1742-6596/2178/1/012035

CFD Flow Engineering. (2023). *Battery Cooling Techniques in Electric Vehicle*. https://cfdflowengineering.com/battery-cooling-techniques-in-electric-vehicle/

Cfd.ninja. (2020). ✅ *Ansys meshing - body of influence - CFD.NINJA*. CFD.NINJA. https://cfd.ninja/ansys-meshing/ansys-meshing-body-of-influence/

CFD Online. (2023). *Pressure based and density based solver -- CFD online discussion forums*. https://www.cfd-online.com/Forums/main/8398-pressure-based-density-basedsolver.html

Eghbalzadeh, A., & Javan, M. (2012). Comparison of mixture and VOF models for numerical simulation of air–entrainment in skimming flow over stepped spillways. *Procedia Engineering*, *28*, 657-660. https://doi.org/10.1016/j.proeng.2012.01.786

Tsavlidis, I. (2023). *Vibration analysis of electric vehicle batteries | blog | SimScale*. SimScale. https://www.simscale.com/blog/vibration-analysis-ev-battery/

Jindal, P., Kumar, B. S., & Bhattacharya, J. (2021). Coupled electrochemical-abuse-heat-transfer model to predict thermal runaway propagation and mitigation strategy for an EV battery module. *Journal of Energy Storage*, *39*, 102619. https://doi.org/10.1016/j.est.2021.102619

Katoch, S. S., & Eswaramoorthy, M. (2020). A detailed review on electric vehicles battery thermal management system. *IOP Conference Series: Materials Science and Engineering*, *912*(4), 042005. https://doi.org/10.1088/1757-899x/912/4/042005

Kim, G., & Pesaran, A. (2007). Battery thermal management design modeling. *World Electric Vehicle Journal*, *1*(1), 126-133. https://doi.org/10.3390/wevj1010126

Lin, Z., Shi, H., Lin, L., Yang, X., Wu, W., & Sun, X. (2021). A high capacity small molecule quinone cathode for rechargeable aqueous zinc-organic batteries. *Nature Communications*, *12*(1). https://doi.org/10.1038/s41467-021-24701-9

Lombardo, T., Duguesnoy, M., El-Bouysidy, H., & Aren, F. (2021). Artificial intelligence applied to battery research: Hype or reality? *Chemical Reviews*. https://doi.org/10.1021/acs.chemrev.1c00108.s001

Moraveji, A., & Toghraie, D. (2017). Computational fluid dynamics simulation of heat transfer and fluid flow characteristics in a vortex tube by considering the various parameters. *International Journal of Heat and Mass Transfer*, *113*, 432-443. https://doi.org/10.1016/j.ijheatmasstransfer.2017.05.095

Morone, P., Cottoni, L., & Giudice, F. (2023). Biofuels: Technology, economics, and policy issues. *Handbook of Biofuels Production*, 55-92. https://doi.org/10.1016/b978-0-32391193-1.00012-3

MR-CFD. (2022). *Introduction to ANSYS fluent: A beginner's guide*. MR CFD. https://www.mrcfd.com/introduction-to-ansys-fluent-a-beginners-guide/

Mutschler, A. (2022). *CFD playing increasing role in design*. Semiconductor Engineering. https://semiengineering.com/cfd-playing-increasing-role-in-design/

Ni, P., & Wang, X. (2020). Temperature field and temperature difference of a battery package for a hybrid car. *Case Studies in Thermal Engineering*, *20*, 100646. https://doi.org/10.1016/j.csite.2020.100646

Nowak, N., Kakade, P. P., & Annapragada, A. V. (2003). Computational fluid dynamics simulation of airflow and aerosol deposition in human lungs. *Annals of Biomedical Engineering*, *31*(4), 374-390. https://doi.org/10.1114/1.1560632

Olabi, A. G., Abbas, Q., Shinde, P. A., & Abdelkareem, M. A. (2023). Rechargeable batteries: Technological advancement, challenges, current and emerging applications. *Energy*, *266*, 126408. https://doi.org/10.1016/j.energy.2022.126408

Rosencrance, L. (2021). *Rosencrance: Time to put the brakes on John A. Charles Jr*. valleytimes.news. https://www.valleytimes.news/opinion/rosencrance-time-to-putthe-brakes-on-john-a-charles-jr/article_cd04a4db-66f3-5516-ada78731afae0e89.html

Shah, R. M., Al Qubeissi, M., Youssef, H., & Soyhan, H. S. (2023). Battery thermal management: An application to petrol hybrid electric vehicles. *Sustainability*, *15*(7), 5868.

https://doi.org/10.3390/su15075868

Simu Tech Group. (2023). *Why is Ansys meshing important for structural FEA and fluid CFD simulations?* SimuTech Group. https://simutechgroup.com/why-is-meshingimportant-for-fea-fluid-simulations/

Tete, P. R., Gupta, M. M., & Joshi, S. S. (2021). Developments in battery thermal management systems for electric vehicles: A technical review. *Journal of Energy Storage*, *35*, 102255. https://doi.org/10.1016/j.est.2021.102255

Teschler, L. (2017). *Efficient cooling for EV battery packs thanks to CFD - Power electronic tips*. Tips on power electronics, FETs, Inverter design and more. https://www.powerelectronictips.com/efficient-cooling-ev-battery-packs-thanks-cfd/

Visontay, E. (2023). *Number of electric vehicles on Australian roads soars as demand exceeds supply*. The Guardian. https://www.theguardian.com/environment/2023/feb/07/number-of-electricvehicles-on-australian-roads-soars-as-demand-exceeds-supply