Simulation and Analysis of Battery using ANSYS FLUENT

submitted in partial fulfillment of

requirements under Industrial training

for the award of

Bachelor of

Technology in

Mechanical Engineering

by

Rayala Lalith Sai Madhav (Regd. No: 18131A03F1)

Routhu Sai Dileep Kumar (Regd. No: 18131A03F7)

Muppidi Abhishek (Regd. No: 18131A03C0)

under the guidance of

Mr. Y. Datta Bharadwaz

Assistant Professor, M.Tech, (Ph.D.)

Department of Mechanical Engineering

GAYATRI VIDYA PARISHAD COLLEGE OF ENGINEERING
(AUTONOMOUS)

(Affiliated to J.N.T.U, Kakinada)

VISAKHAPATNAM - 530048

CERTIFICATE

This is to certify that the project titled **CFD ANALYSIS OF EV BATTERY USING ANSYS FLUENT** is a bonafide record of the work done by

RAYALA LALITH SAI MADHAV (Regd. No: 18131A03F1)
ROUTHU SAI DILEEP KUMAR (Regd. No: 18131A03F7)
MUPPIDI ABHISHEK (Regd. No: 18131A03C0)

in partial fulfillment of the requirements for the award of the degree of

Bachelor of Technology in Mechanical Engineering

of the Gayatri Vidya Parishad College of Engineering (Autonomous) affiliated to

Jawaharlal Nehru Technological University, Kakinada.

Mr. Y. Datta Bharadwaz
Assistant Professor
Dept. of Mechanical Engineering
Gayatri Vidya Parishad College
Of Engineering (A)

Dr. B. Govind Rao
Professor and Head of the Department
Dept. of Mechanical Engineering
Gayatri Vidya Parishad College
Of Engineering (A)

ACKNOWLEDGEMENTS

I would like to thank **Dr. B. Govinda Rao** (Head of Department), Gayatri Vidya Parishad College of Engineering (Autonomous), for giving me this opportunity to learn and enhance my soft skills.

Sincere regards and thanks to **Mr. Y. Datta Bharadwaz,** Assistant Professor, Gayatri Vidya Parishad College of Engineering (Autonomous), who gave me constant motivation for enhancing my knowledge and skills.

I express my sincere gratitude to **INTERNSHALA TRAININGS** for having opened up the avenues to undertake this remote **ANSYS** Training.

I hope that my training project report will reflect my technical knowledge and innovations, which I gained from **INTERNSHALA TRAININGS** and **VERZEO Edu-Tech.**

ABSTRACT

In view of the projected global energy demand and increasing levels of greenhouse gases (CO2) and pollutants (NOx, SOx, and fine particulates) there is a great need for new energy sources and technologies which provide clean and environmentally friendly solutions to the requirements of the end-users. The purpose of a battery thermal management system (BTMS) is to maintain the battery safety and efficient use as well as ensure the battery temperature is within the safe operating range.

Increase in heat generation inside a battery is desirable for the optimal functioning of the battery. However, providing cooling for the battery can reduce these heat build-ups inside the battery, as these unwanted heat build-ups can cause reduced discharging and charging rates, reduced life of the battery, and thus, in a way, responsible for lesser efficiency of the battery. Thermal modeling was performed to study the effect of the electrode configuration on the thermal behavior of a lithium-polymer battery. It was examined the effect of the configuration of the electrodes such as the aspect ratio of the electrodes and the placing of current collecting tabs as well as the discharge rates on the thermal behavior of the battery. The potential and current density distribution on the electrodes of a lithium-polymer battery were predicted as a function of discharge time by using the finite element method. Then, based on the results of the modeling of potential and current density distributions, the temperature distributions of the lithium-polymer battery were calculated. The temperature distributions from the modeling were in good agreement with those from the experimental measurement for the batteries.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

CHAPTER – 1

ANSYS

1.1 INTRODUCTION

Ansys is an American company based in Canonsburg, Pennsylvania. It develops and markets CAE/multi-physics engineering simulation software for product design, testing and operation and offers its products and services to customers worldwide.

Ansys was founded in 1970 by John Swanson, who sold his interest in the company to venture capitalists in 1993. Ansys went public on NASDAQ in 1996. In the 2000s, the company acquired numerous other engineering design companies, obtaining additional technology for fluid dynamics, electronics design, and physics analysis. Ansys became a component of the NASDAQ-100 index on December 23, 2019.

1.2 ENGINEERING SIMULATION SOFTWARE

Ansys develops and markets engineering simulation software for use across the product life cycle. Ansys Mechanical finite element analysis software is used to simulate computer models of structures, electronics, or machine components for analyzing strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other attributes. Ansys is used to determine how a product will function with different specifications, without building test products or conducting crash tests. For example, Ansys software may simulate how a bridge will hold up after years of traffic, how to best process salmon in a cannery to reduce waste, or how to design a slide that uses less material without sacrificing safety.

Most Ansys simulations are performed using the Ansys Workbench system, which is one of the company's main products. Typically, Ansys users break down larger structures into small components that are each modelled and tested individually. A user may start by defining the dimensions of an object, and then adding weight, pressure, temperature and other physical properties. Finally, the Ansys software simulates and analyzes movement, fatigue, fractures, fluid flow, temperature distribution, electromagnetic efficiency and other effects over time.

Ansys also develops software for data management and backup, academic research and teaching.

1.3 HISTORY

The first commercial version of Ansys software was labelled version 2.0 and released in 1971. At the time, the software was made up of boxes of punch cards, and the program was typically run overnight to get results the following morning. In 1975, non-linear and thermo-electric features were added. The software was exclusively used on mainframes, until version 3.0 (the second release) was introduced for the VAX station in 1979. Version 3 had a command line interface like DOS.

In 1980, Apple II was released, allowing Ansys to convert to a graphical user interface in version 4 later that year. Version 4 of the Ansys software was easier to use and added features to simulate electromagnetism. In 1989, Ansys began working with Compuflo. Compuflo's Flotran fluid dynamics software was integrated into Ansys by version 5, which was released in 1993. Performance improvements in version 5.1 shortened processing time two to four-fold and was followed by a series of performance improvements to keep pace with advancements in computing. Ansys also began integrating its software with CAD software, such as Autodesk.

In 1996, Ansys released the Design-Space structural analysis software, the LS-DYNA crash and drop test simulation product, and the Ansys Computational Fluid Dynamics (CFD) simulator. Ansys also added parallel processing support for PCs with multiple processors. Version 8.0 was published in 2005 and introduced Ansys' fluid–structure interaction software, which simulates the effect structures and fluids have on one another. Ansys also released its Probabilistic Design System and DesignXplorer software products, which both deal with probabilities and randomness of physical elements.

Version 15 of Ansys was released in 2014.[36] It added a new features for composites, bolted connections, and better mesh tools.[36] In February 2015, version 16 introduced the AIM physics engine and Electronics Desktop, which is for semiconductor design. Version 18 allowed users to collect real-world data from products and then incorporate that data into future simulations.

1.4 PRODUCTS

Ansys has created and assembled a suite of software programs that span the needs of designers to analysts. And these are the program products offered by Ansys:

- Fluids. Ansys Blade-Modeler. Ansys CFD Enterprise.
- Structures. Ansys Mechanical Enterprise. Ansys Mechanical Premium.
- Electromagnetics. Ansys Electronics Desktop.
- Semiconductors. Ansys Path FX.
- Embedded Software. Ansys SCADE Architect.
- Platform. Ansys Cloud.
- 3D Design. Ansys Discovery AIM.

CHAPTER 2 - ANSYS FLUENT

Ansys Fluent gives you more time to innovate and optimize product performance. Trust your simulation results with a software that has been extensively validated across a wide range of applications. With Ansys Fluent, you can create advanced physics models and analyze a variety of fluids phenomena—all in a customizable and intuitive space.

Ansys Fluent unlocks new potentials for CFD analysis. A fluid simulation software with fast pre-processing and faster solve times to help you be the fastest to break into the market. Fluent's industry leading features enable limitless innovation, while never making a compromise on accuracy.

Key Features

It has the best fluid simulation tools on the market in an efficient, customizable workspace:

- Streamlined Workflows
- Battery Modeling
- Electric Motor Cooling
- Turbulence Modeling
- Multiphase Flows
- Combustion Models

Ansys Fluent is a fluid simulation program that is known for its advanced physics modeling capabilities and accuracy.

- User-Friendly Interface.
- Best-In-Class Physics Models.
- Single Window, Streamlined Workflow.
- Parallel Capabilities for Meshing and Solving

Like all finite element software, it really depends on what you want to simulate. For mechanical and multi-physics problems I found that in Ansys it is very easy to link different simulations. Another thing is that there are a lot of mesh options that allow you to control your mesh and subsequent elements to a high degree. The mesh options combined with the contact options allow for easy modeling of seemingly complex geometry in an easy and structured way. The contact definition allows you to connect parts with different nodal interfaces (in the previous software used at work the mesh had to match at part interfaces adding a level of complexity to the mesh generation).

ANSYS FLUENT is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries. ANSYS FLUENT is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control is all possible. In addition, ANSYS FLUENT uses a client/server architecture, which enables it to run as separate simultaneous processes on client desktop workstations and powerful compute servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems. ANSYS FLUENT provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes. ANSYS FLUENT also enables you to refine or coarsen your mesh based on the flow solution. After a mesh has been read into it, all remaining operations are performed within ANSYS FLUENT. These include setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and postprocessing and viewing the results. The ANSYS FLUENT serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer. ANSYS FLUENT also uses a utility called cortex that manages it’s user interface and basic graphical functions. Its parallel solver enables you to compute a solution using multiple processes that may be executing on the same computer, or on different computers in a network. Parallel processing in it involves an interaction between ANSYS FLUENT, a post-process, and a set of compute-node processes. It interacts with the host process and the collection of compute nodes using the cortex user interface utility.

CHAPTER 3 - SIMULATION, ANALYSIS AND THERMAL COOLING OF BATTERY CELL

3.1 Introduction to battery cell

Firstly, a battery cell consists of a single cell arrangement of cell body, a cathode terminal, and an anode terminal. This is the basic construction involved in any battery in fact. We have considered the working and analysis of a battery whose dimensions and constants are derived from a research paper from a reputed firm, which highly focussed on the similar working of such battery.

Any battery contains electrons flowing through it when the circuit is closed, thus generating electricity in the circuit. This flow of electrons is caused due to the available change in potential between the two oppositely charged terminals. This is considered as the “Discharging” reaction.

On the other hand, while “charging” occurs, electrons travel in the opposite path as potential is applied in the opposite reaction. This cycle of charging and discharging of (any) battery causes a rapid flow of electrons within the battery, thereby generating more kinetic energy that is held by the electrons. Major amount of this energy is emitted out of the electrons in the form of heat energy, by the electrons. With daily use and over the passage of time, heat energy accumulates and highly affects the functioning of the battery. Thus, proper thermal cooling of the battery is an essential element, required for the optimum performance of any battery.

3.2 Understanding the battery model in Ansys-Fluent

Ansys battery modeling and simulation solutions use multi-physics to help you maximize battery performance and safety while reducing cost and testing time. For the battery taken into consideration, we use the MSMD Battery Model present in the ANSYS Fluent Models Database. Prior to beginning the simulation of the battery, accurate dimensions of the battery cell are to be finalized. We adopted the model of a battery extensively used in a highly reputed research paper on **“U. S. Kim et al, "Effect of electrode configuration on the thermal behavior of a lithium-polymer battery", Journal of Power Sources, Volume 180 (2), pages 909-916, 2008.”** And the module in Ansys fluent, used for analysis is **MSMD BATTERY MODEL** and **NTGK** method sub model.

3.3 Schematic diagram, geometry and dimensions of the battery cell:

The following are the diagrams representing the Schematic, geometric and dimensional properties respectively, as shown below.

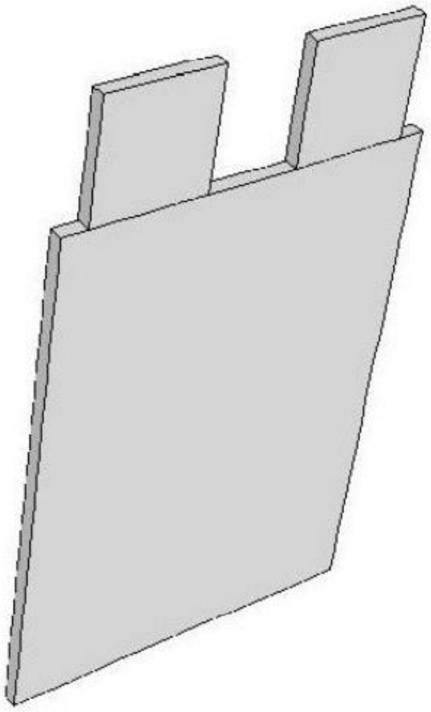

Fig. 3.1 Schematic Diagram of cell

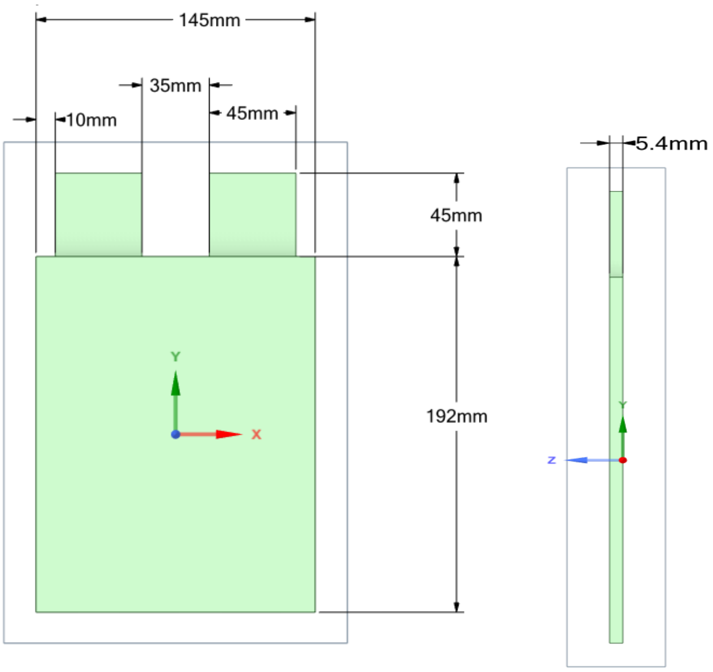

Fig. 3.2 Geometry and Dimensions of each cell

3.4 Meshing: After generating the geometry in Design Modeller, we performed the meshing operation for the imported geometry model.

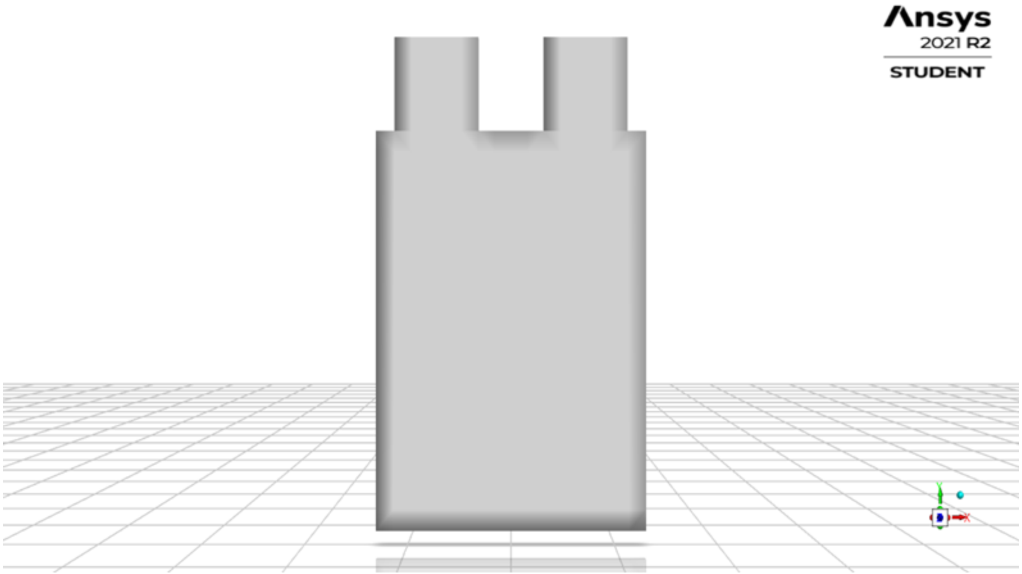

Fig. 3.3 Component after generating the mesh.

Zones and Components: The cell body acts as an active zone (and the material used is e-material).The terminals, anode, and cathode act as the passive zones (with n-material and p-material respectively.

A) Simulating a single cell battery using the MSMD battery model (without cooling)

Set up a battery cell simulation using the NTGK battery sub-model

Perform the calculations for different battery discharge rates and compare the results using the postprocessing capabilities of Ansys Fluent.

Basic procedure involved after generating geometry and mesh:

1. Setting the general settings.
2. Selecting and Enabling the Battery model and energy model.
3. Create suitable materials for cell-body, cathode, and anode.
4. Give Cell zone conditions, i.e., assign these materials to respective sites
5. Provide necessary boundary conditions.
6. Give necessary input regarding the report definitions, report plots, and residuals that we obtain during post-processing.
7. Initialize the problem.
8. Set up an optimal number of timesteps and iterations and run calculations.
9. After calculations are processed, observe and take the residual graphs obtained in the analysis.
10. Display contours of static temperature, potential variation in the cell, vectors of current density, etc, to get a detailed understanding of the battery cell taken.

3.5 Battery Model

The first thing involved in setting up the model is turning ON the pre-installed "Battery Model". The following dialog box appears as shown in Fig. 4. Select the MSMD solution method and the NTGK empirical model for solving. Specify C-rate as the default value '1'. Also, retain the capacity as 14.6 A-h.

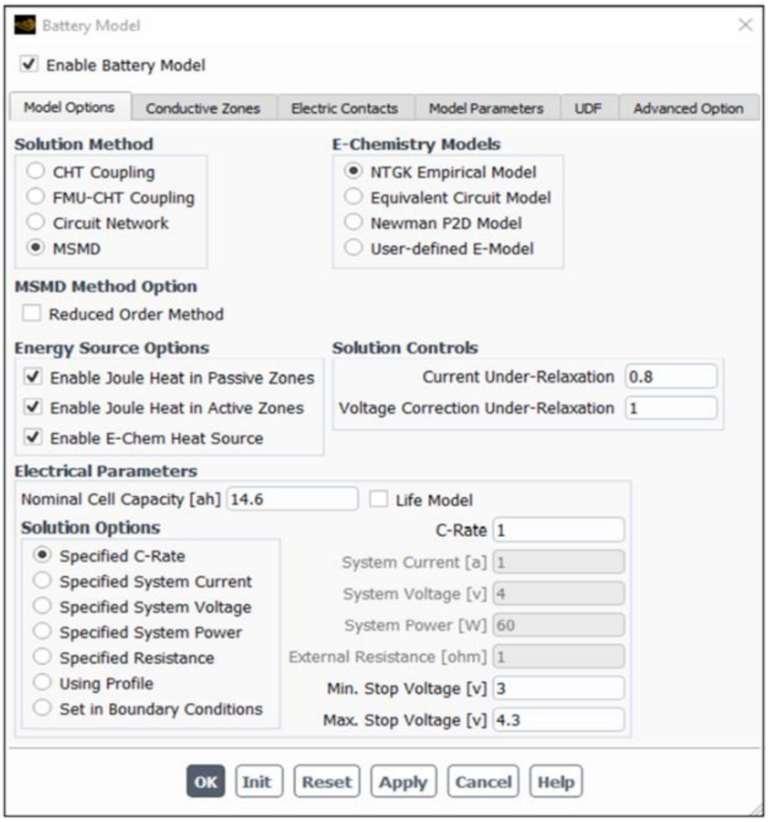

Fig. 3.4 Enabling battery model

After setting the Model Options, move to the Conductive zones tab. In this tab, we shall give the active and passive components as per the battery geometry. Zones are selected as shown in Fig. 3.5.

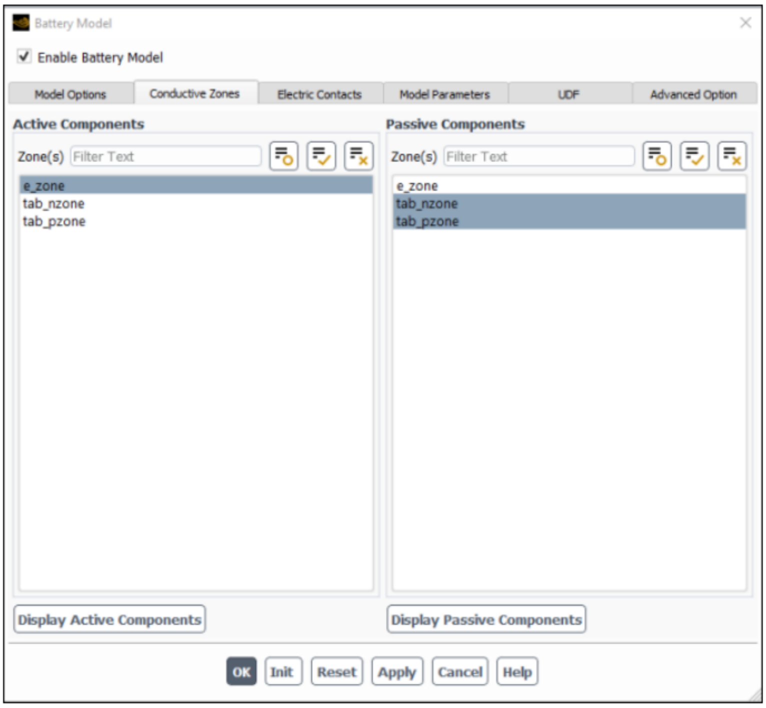

Fig. 3.5 Assigning and selecting conductive zones

Group	Control or List	Value or Selection
Active Components	Zone (s)	e-zone
Passive Components	Zone (s)	tab_nzone
		tab_pzone

Table 3.1 Conductive zones for cell

Group	Control or List	Value or Selection
External Connectors	Negative Tab	tab_n
	Positive Tab	tab_p

Table 3.2 Electric contacts for cell

After assigning the conductive zones respectively, open the electric contacts tab to assign the external connectors of the battery as in Fig. 3.6.

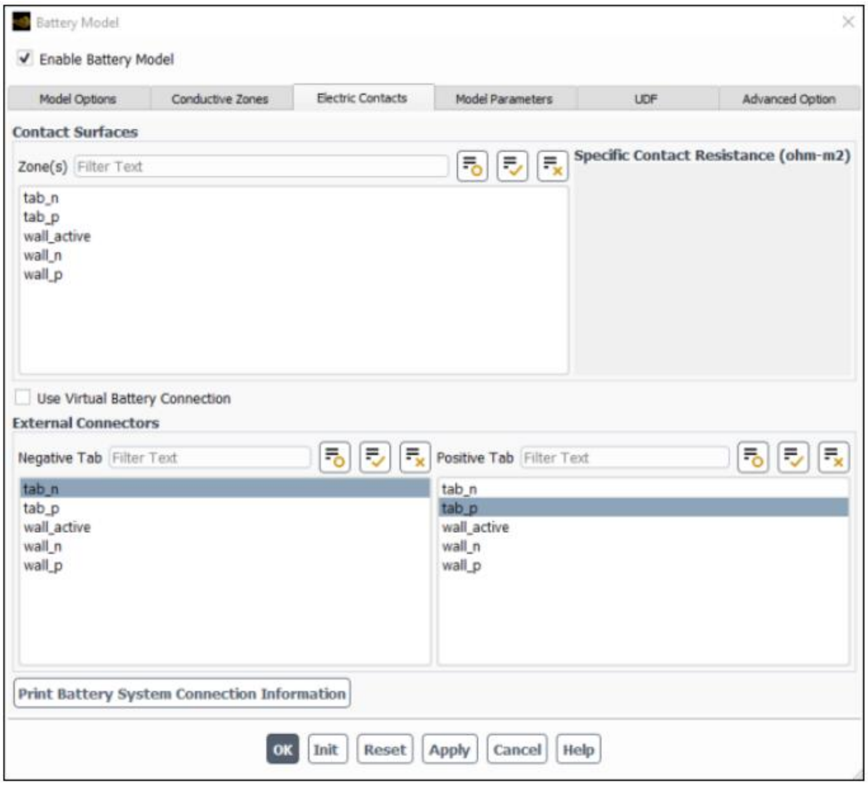

Fig. 3.6 Electric Contacts

```
Battery Network Zone Information:
-------------------------------------
 Battery Serial 1
   Parallel 1
         Active zone:  e_zone
-----------------------------------

 Passive zone 0:
        tab_nzone
 Passive zone 1:
        tab_pzone

Number of battery series stages =1;  Number of batteries in parallel per series stage=1
****************END OF BATTERY CONNECTION INFO**************
```

Fig. 3.7 Battery system information in the console

After assigning the electric contacts, print the battery information to the console, to check for any unexpected errors in assigning.

3.6 Material settings

After enabling the energy model and battery model, we move to create the materials required for the battery components as seen in Fig. 3.8 and 3.9.

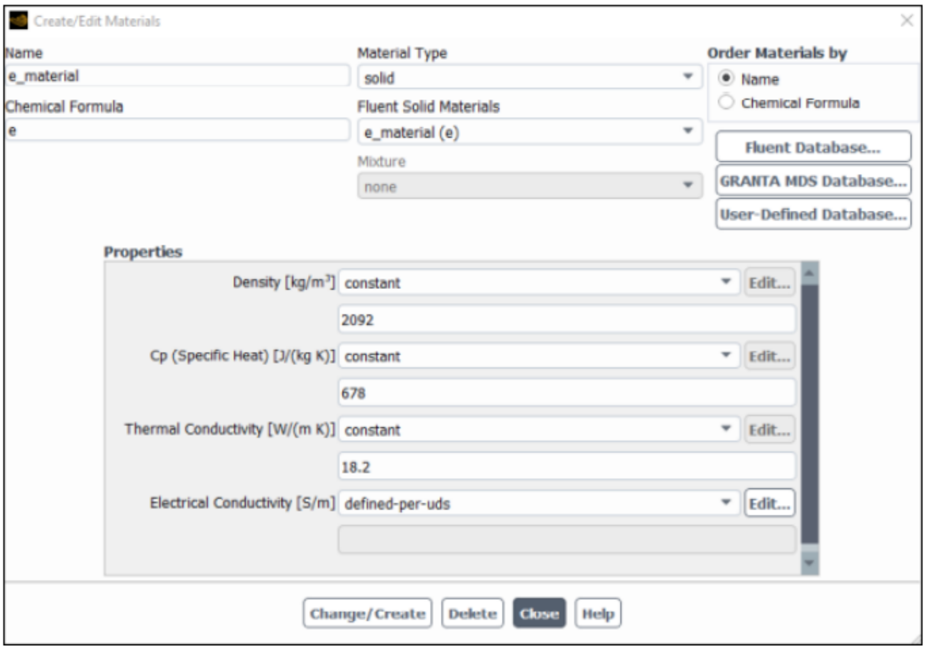

Fig. 3.8 Creating Materials (active material)

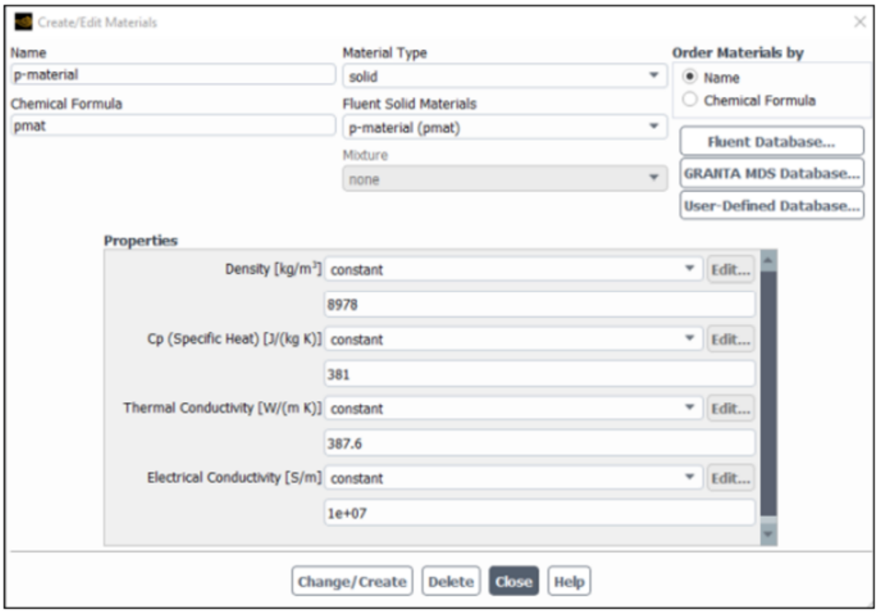

Fig. 3.9 Passive Material (for anode and cathode)

Cell-zone conditions are given as per the battery's configuration. Each zone is assigned with its corresponding material type.

Next Boundary conditions are to be given. We consider the Convection model in the Thermal tab and set the free stream temperature as 300K and heat transfer coefficient as 0 to restrict the heat transfer first, and then provide heat transfer coefficient as 20, as forced convection.

After assigning the boundary conditions, we move on to the Solution settings.

We first turn off the turbulence and flow equations. Later, we give definitions of the reports required while processing the given conditions model. We give information regarding the residual plots and other report plots, as we can see further.

After giving all the necessary conditions and report definitions, we perform initialization using the Standard Initialization command.

Next, to run the calculation, we set the optimal required number of timesteps and iterations and then click on Calculate.

After the calculation is completed, the following residuals and results are obtained.

Other contours are obtained through contours and vectors commands in the Results tab.

3.7 Solutions and graphs:

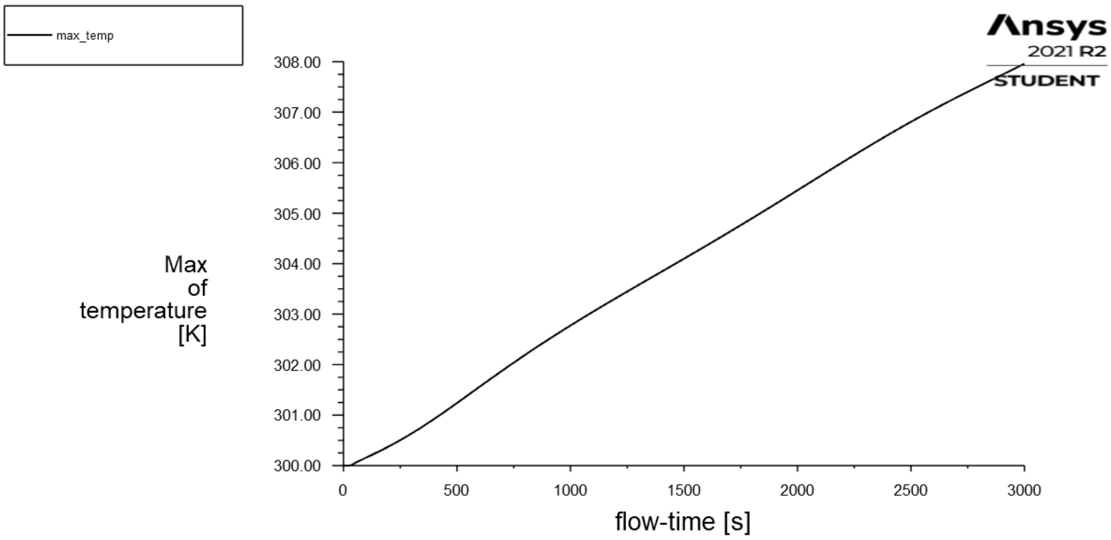

Fig. 3.10 Max-temperature plot

It is observed that with the increase in time, the temperature of the cell keeps on increasing, as there is no possibility of heat transfer to the surroundings.

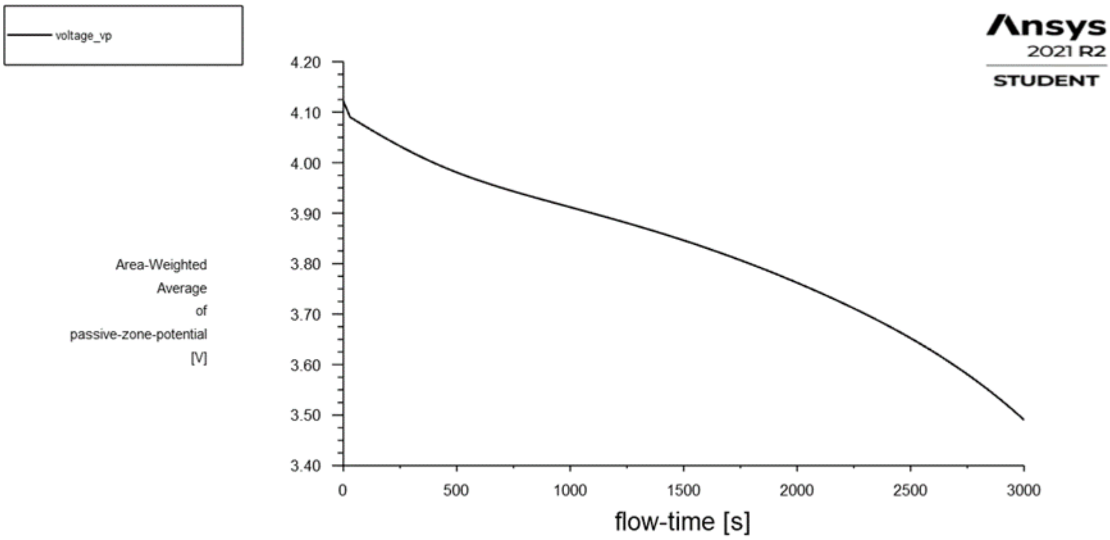

Fig. 3.11 Passive zone potential plot

It is seen that with the increase in time, the passive zone or the terminal potential keeps reducing.

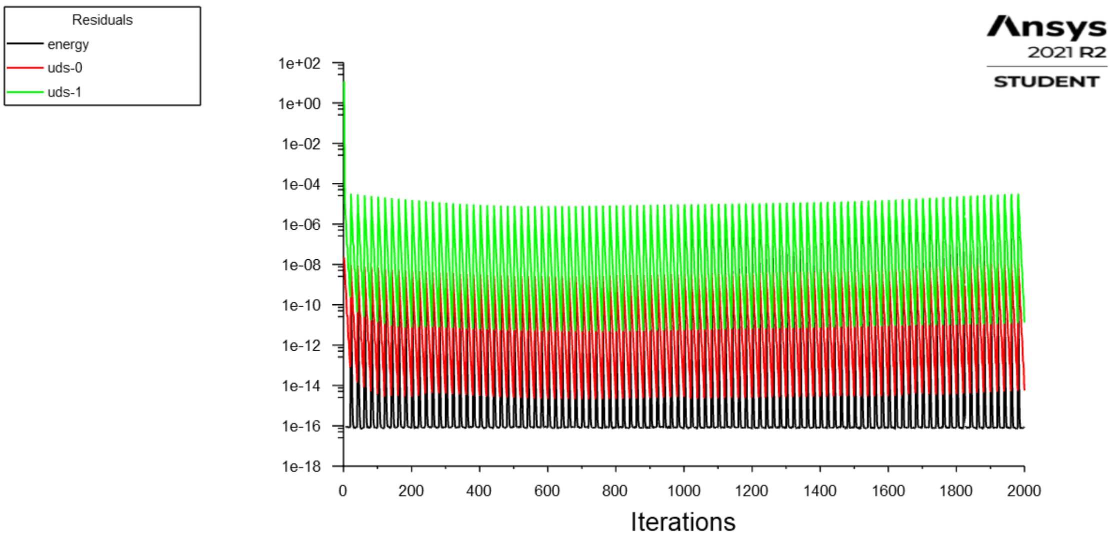

Fig. 3.12 Scaled residuals plot

Here, uds0 corresponds to the positive potential, and uds1 corresponds to the negative potential

Post-processing results

Contours and vector plots as shown are obtained by selecting the suitable zones and setting the required variables in the contours display window dialog box and vector display dialog box.

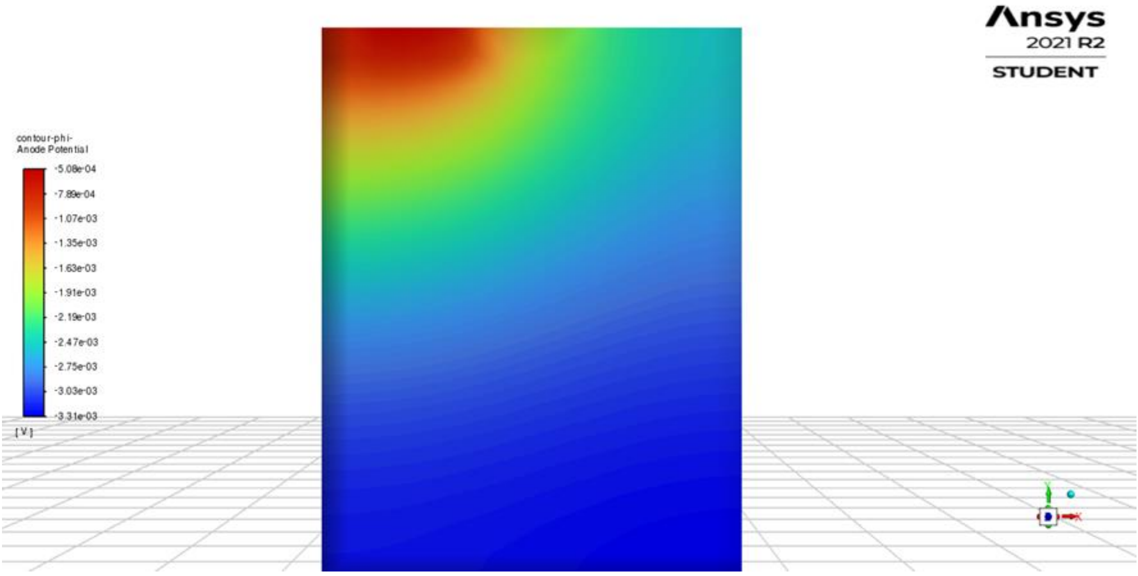

Fig. 3.13. Anode Potential Contour without cooling. The variation of anode potential is seen. The magnitude decreases from the top to a minimum at the bottom.

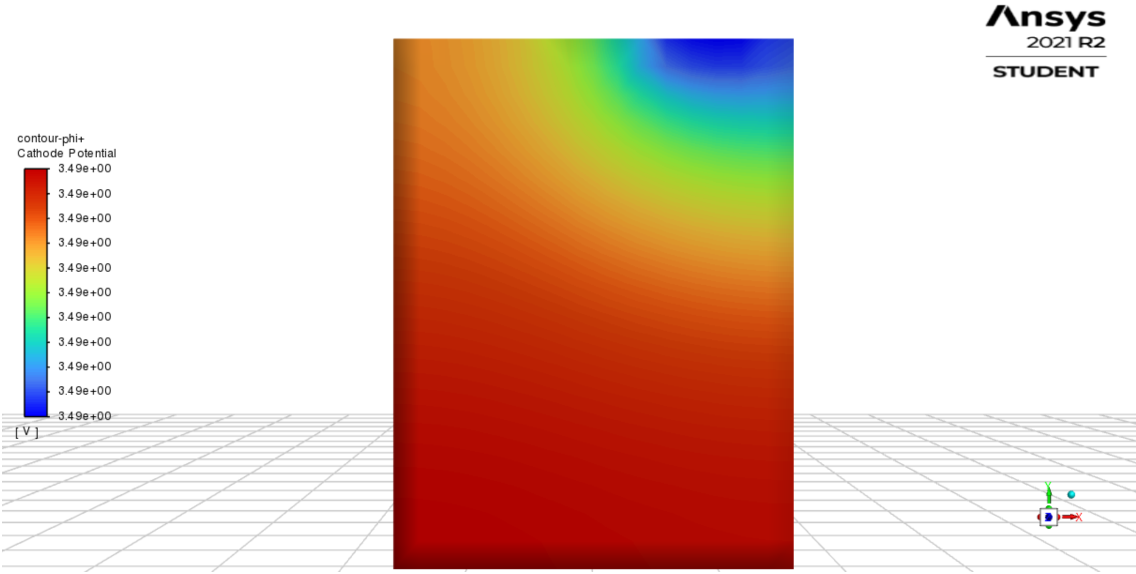

Fig. 3.14. Cathode potential contour without cooling

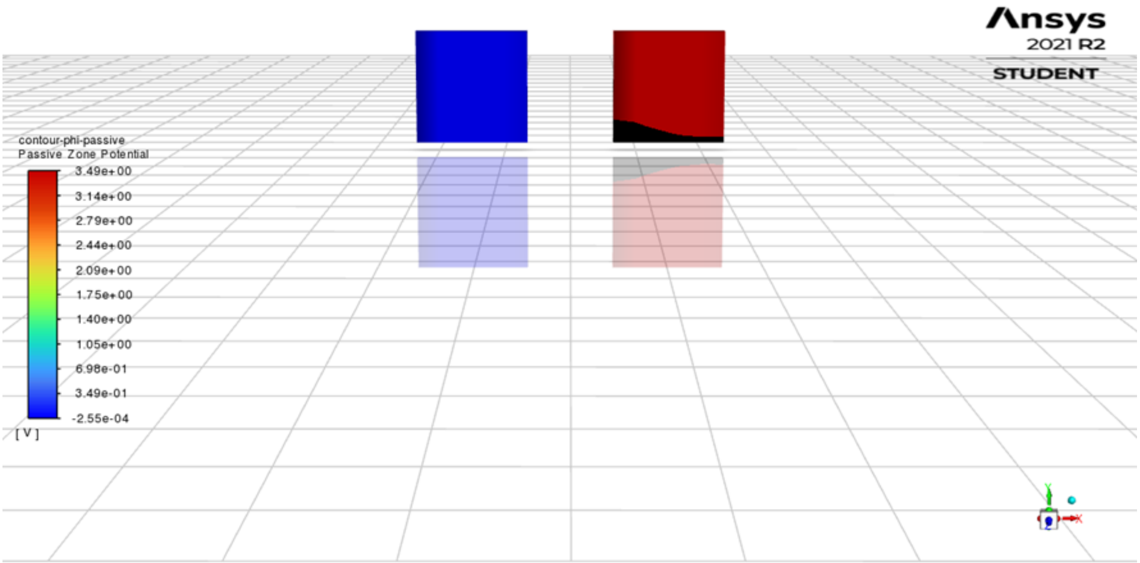

Fig. 3.15. Passive zone potential contour without cooling

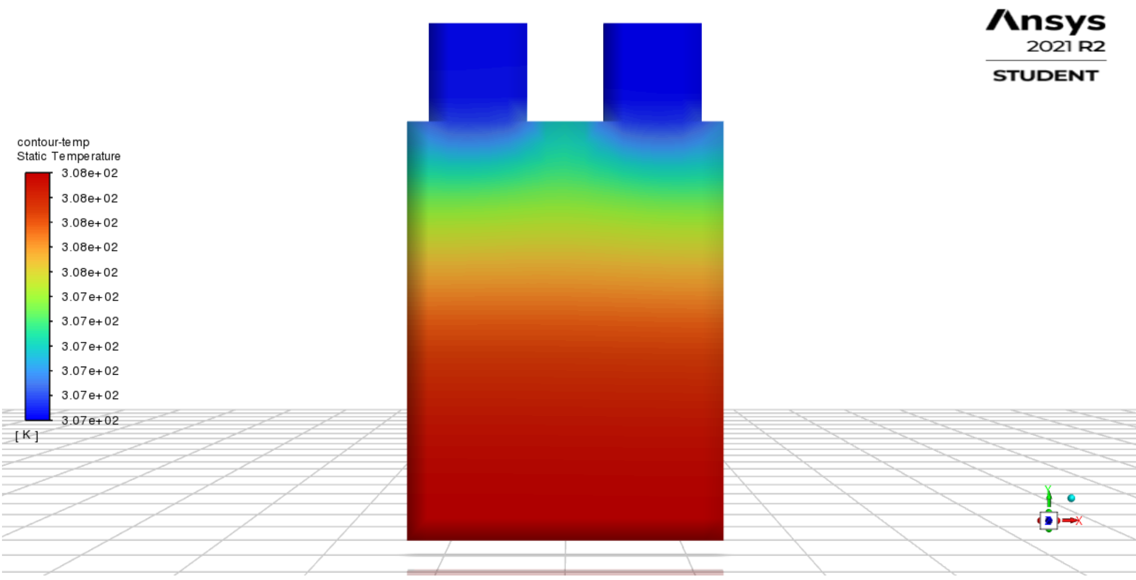

Fig. 3.16. Static temperature contour, without cooling condition.

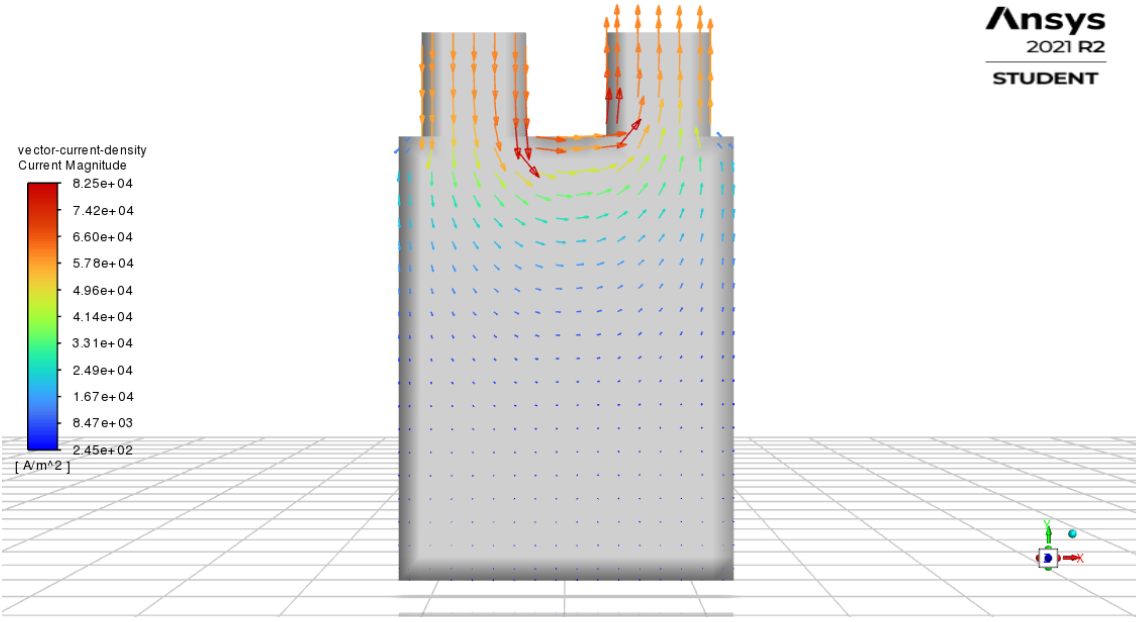

Fig. 3.17. Current density vectors, without cooling condition.

This shows the flow of current vectors from the anode to the cathode. The density is higher near the terminals and least in the bottom the cell.

B) SIMULATING A SINGLE CELL BATTERY USING THE MSMD BATTERY MODEL (with cooling)

The change in procedure occurs only in the case of assigning boundary conditions. For “without cooling” model, there is no consideration of any heat transfer (h=0), implying no cooling taking place. Here, in “with cooling” model, forced heat convection is taken (h=20), implying transfer of heat to external materials.

3.8 Solutions and graphs (with cooling):

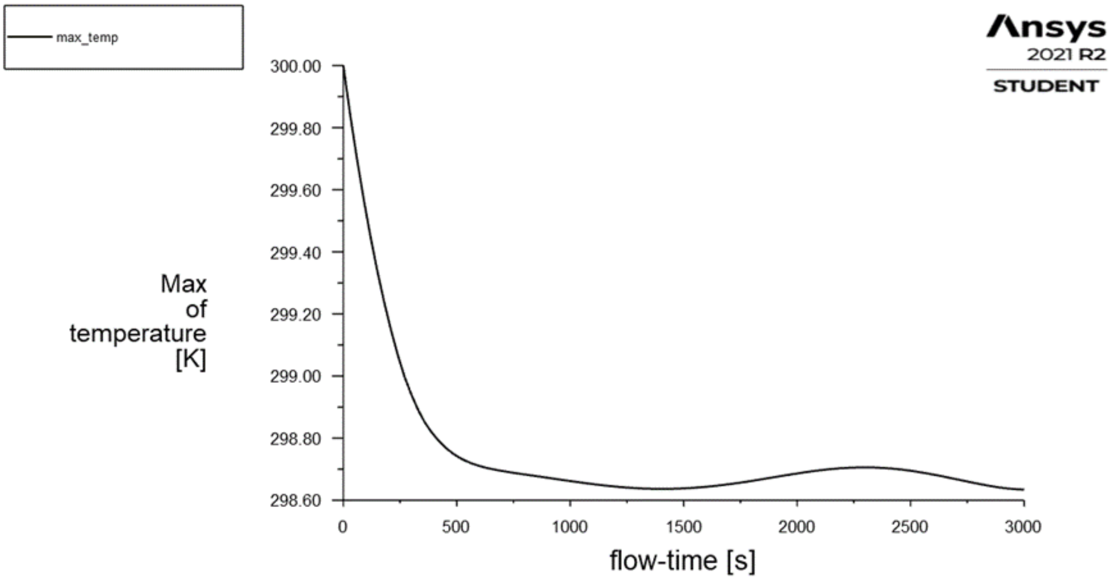

Fig. 3.18. Max-temperature plot (with cooling)

After providing the forced convection for the battery cell, there is a change in temperature variation when compared to that in case of no heat transfer condition.

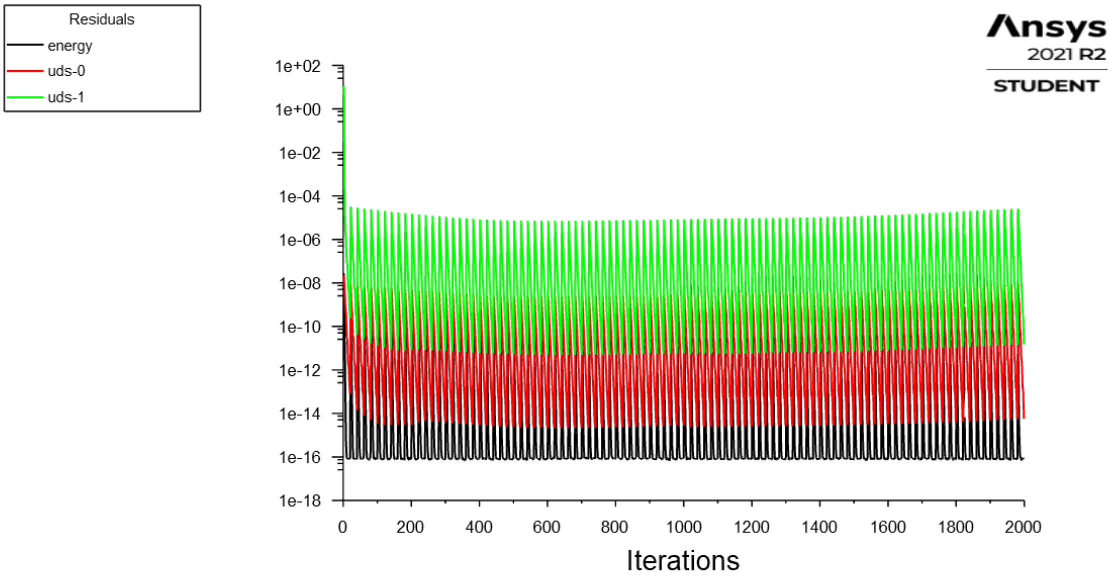

Fig. 3.19. Residuals plot (with cooling)

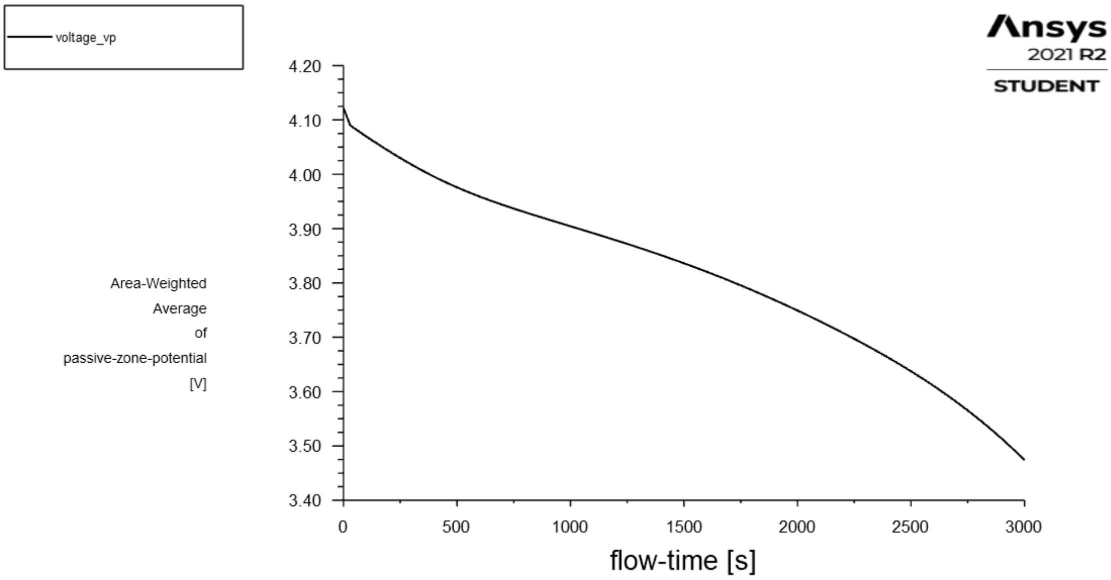

Fig. 3.20. Passive zone potential plot (with cooling)

Post processing results (with cooling)

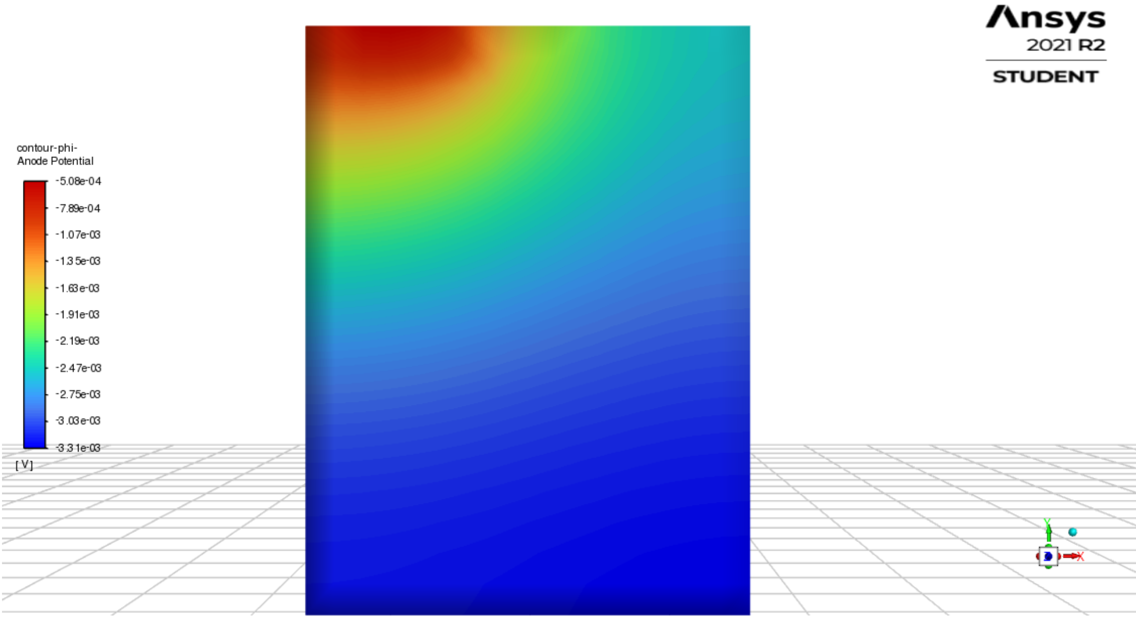

Fig. 3.21. Anode Potential contour (with cooling)

It is similar to the one in the no convection model, however, there is a change in the magnitude of the potential.

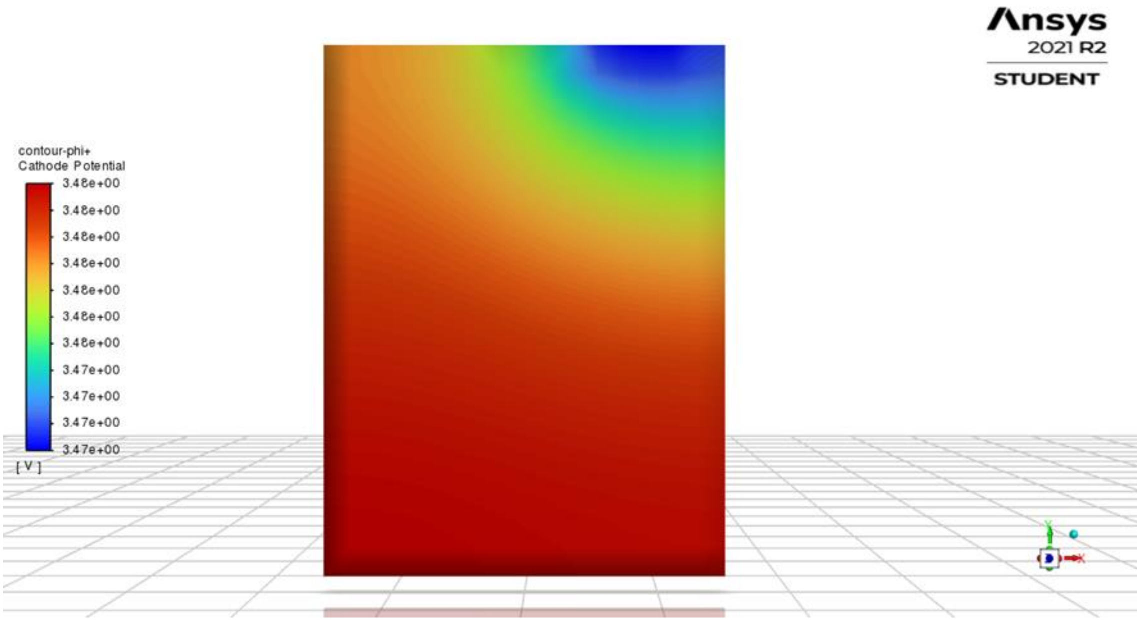

Fig. 3.22. Cathode Potential contour (with cooling)

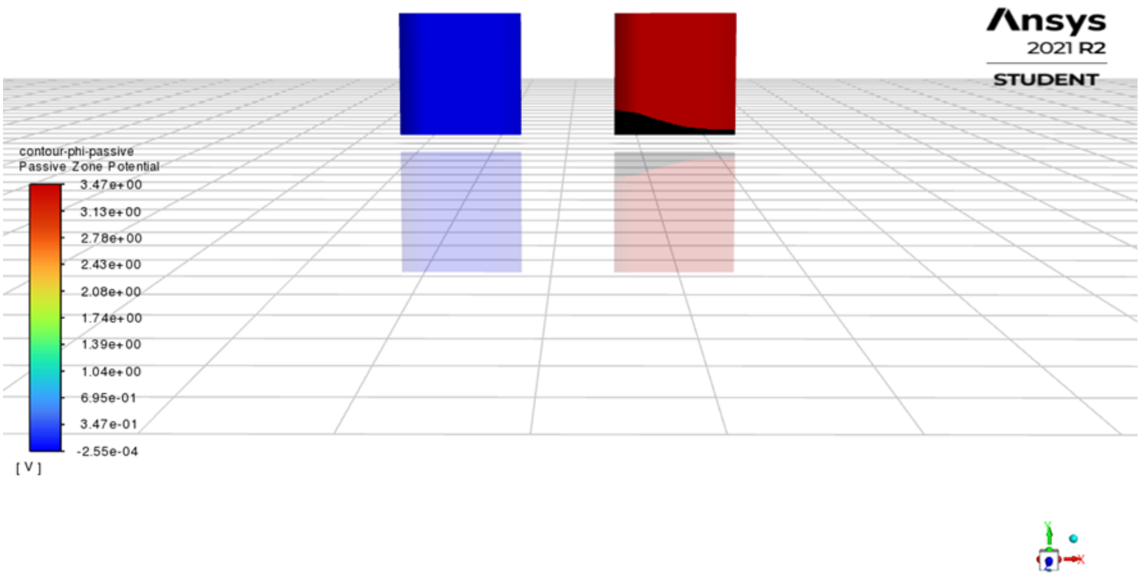

Fig. 3.23. Passive zone potential contour (with cooling)

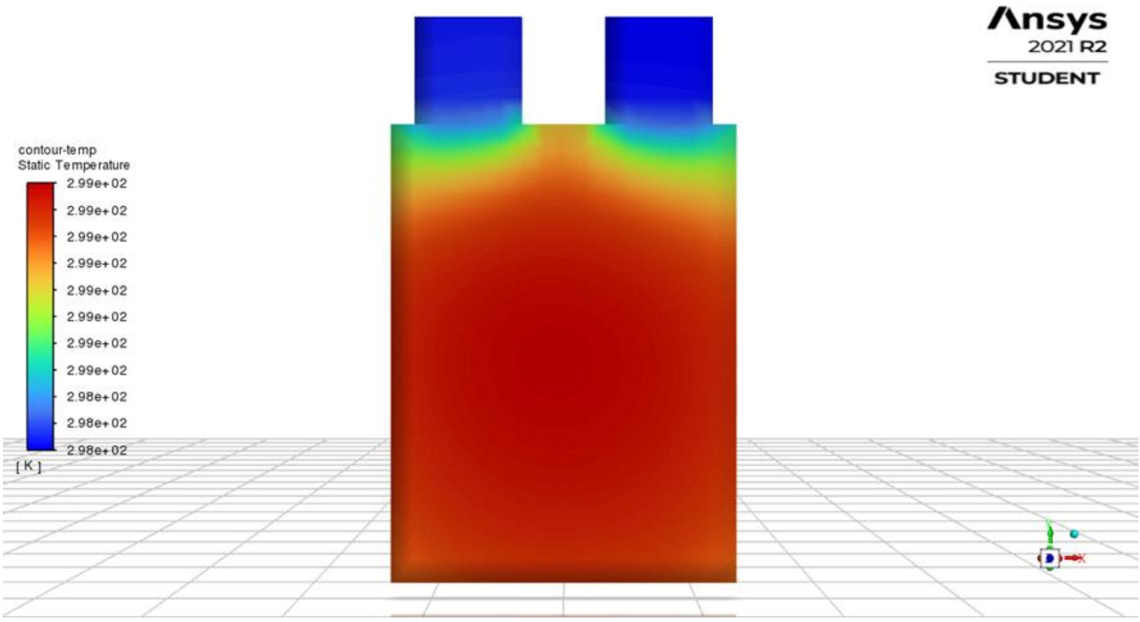

Fig. 3.24. Static temperature contour (with cooling)

This shows the variation of temperature along the body of the batter cell.

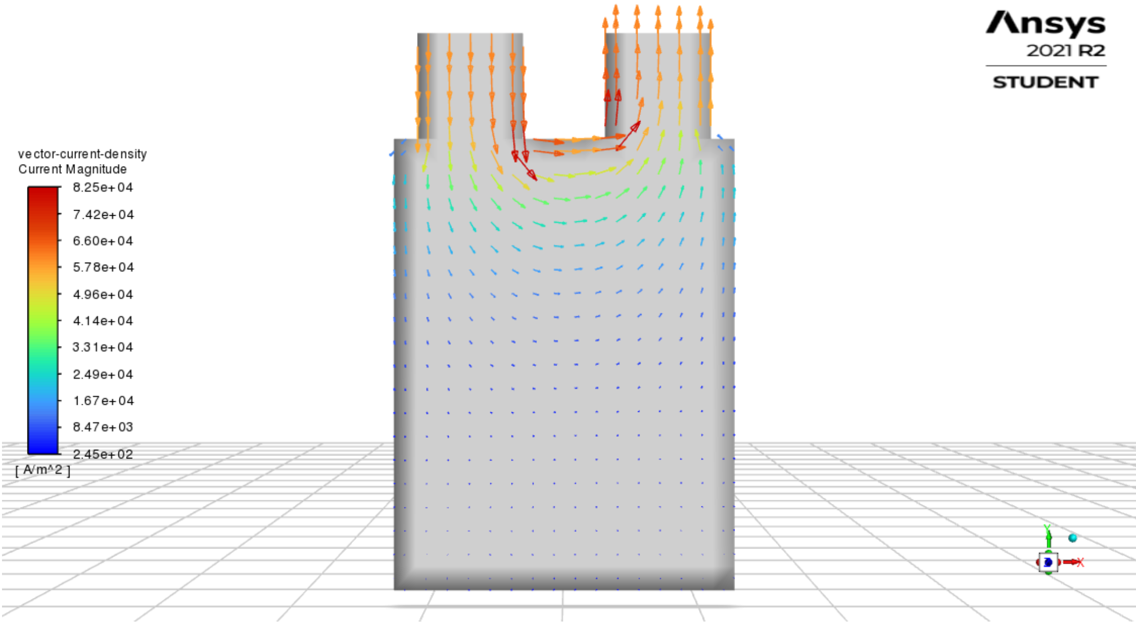

Fig. 3.25. Vectors of current density (with cooling condition)

It is seen that after providing cooling, the magnitude of electric current density has increased. This proves the claim that reducing the heat accumulation through forced convection, makes the battery to perform better and deliver more power output, thereby increasing the battery life and efficiency.

CHAPTER - 4 BATTERY PACK MODEL

SIMULATING A 1P3S BATTERY PACK USING THE BATTERY PACK

The procedure, geometry, and dimensions of each individual cells of the battery pack are the same. We have assembled 3 such similar battery cells and joined them with help of busbars, to perform real-time application analysis, as batteries are always in stacks of induvial cells connected together for more output.

4.1 Schematic diagram

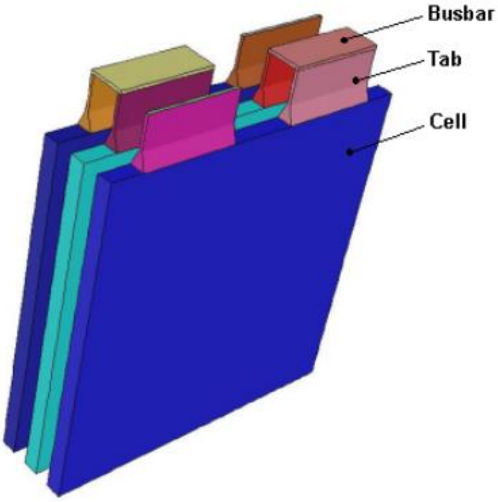

Fig. 4.1 Battery pack schematic.

The geometry and dimensions of each cell taken in the pack are as per Fig. 2 and Fig. 3.

4.2 Meshing the battery pack

After modeling in Design Modeller, meshing is done as shown. In Fig. 24.

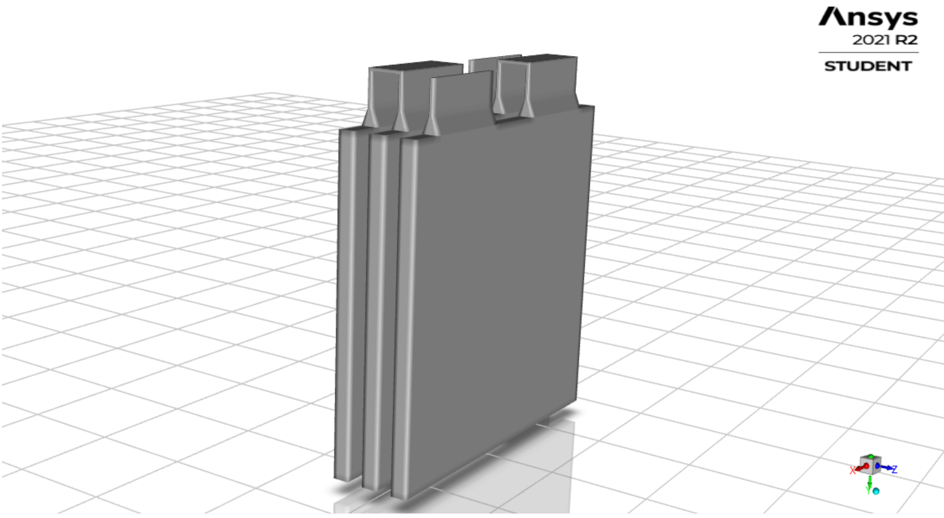

Fig. 4.2 Battery pack mesh

Group	Control or List	Value or Selection
Active Components	Zone (s)	cell_1
		cell_2
		cell_3
Passive Components	Zone (s)	n_tabzone_1
		n_tabzone_2
		n_tabzone_3
		p_tabzone_1
		p_tabzone_2
		p_tabzone_3
		bar1
		bar2

Table – 4.1 Conductive zones of battery pack

Group	Control or List	Value or Selection
External Connectors	Negative Tab	tab_n
	Positive Tab	tab_p

Table – 4.2 Electric contacts for battery pack

A) SIMULATING A 1P3S BATTERY PACK USING THE BATTERY MODEL (without cooling)

The process used for cell with h=0 is repeated for the pack as well and the results are obtained.

4.3 Solutions and graphs (without cooling condition):

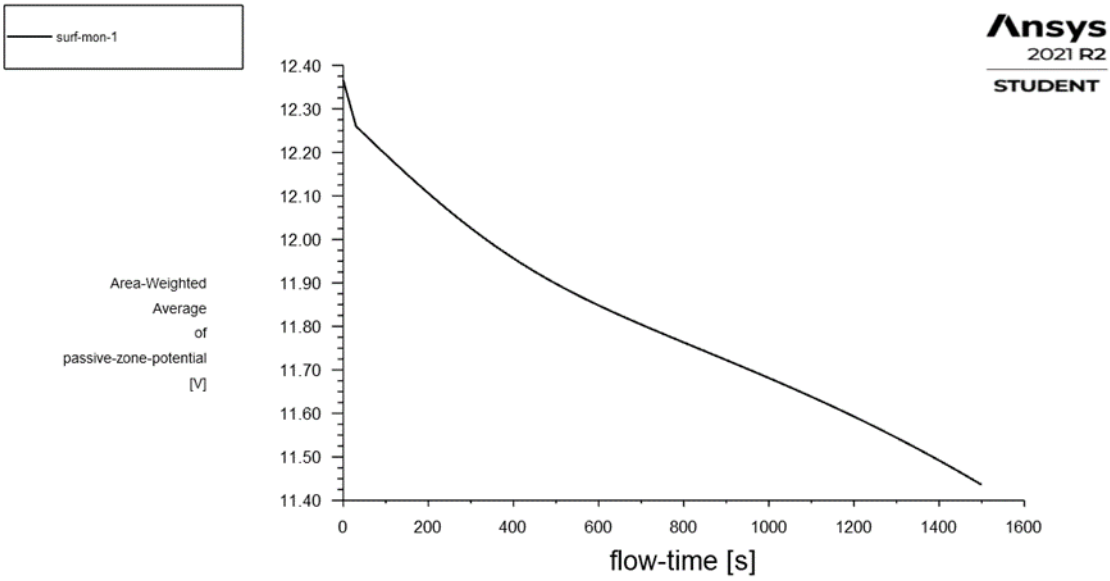

Fig. 4.3 Passive zone potential plot (without cooling)

This is similar to what is obtained in the single battery cell.

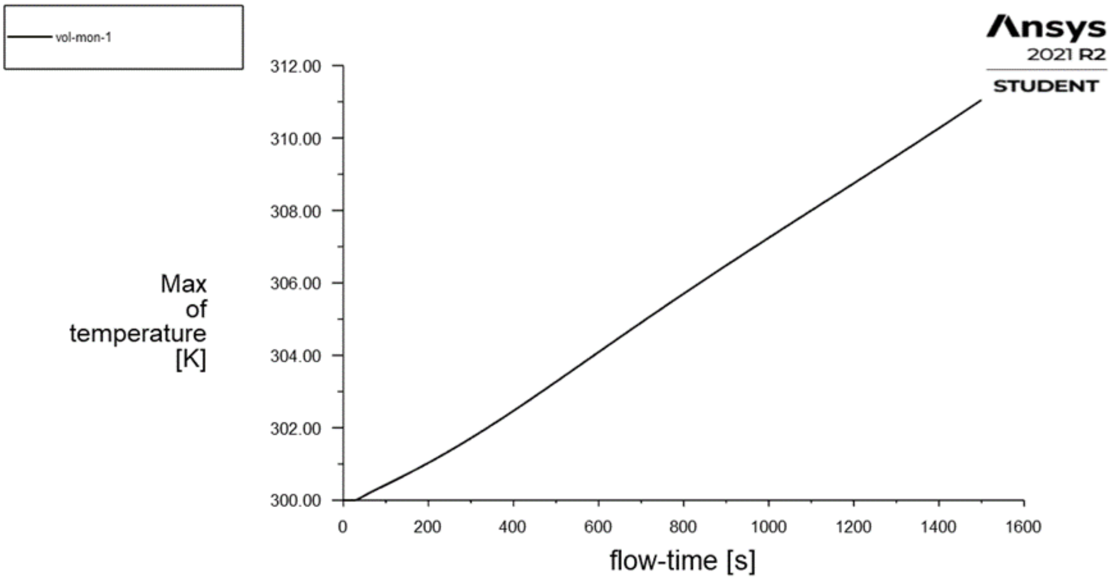

Fig. 4.4 Max temperature plot (without cooling)

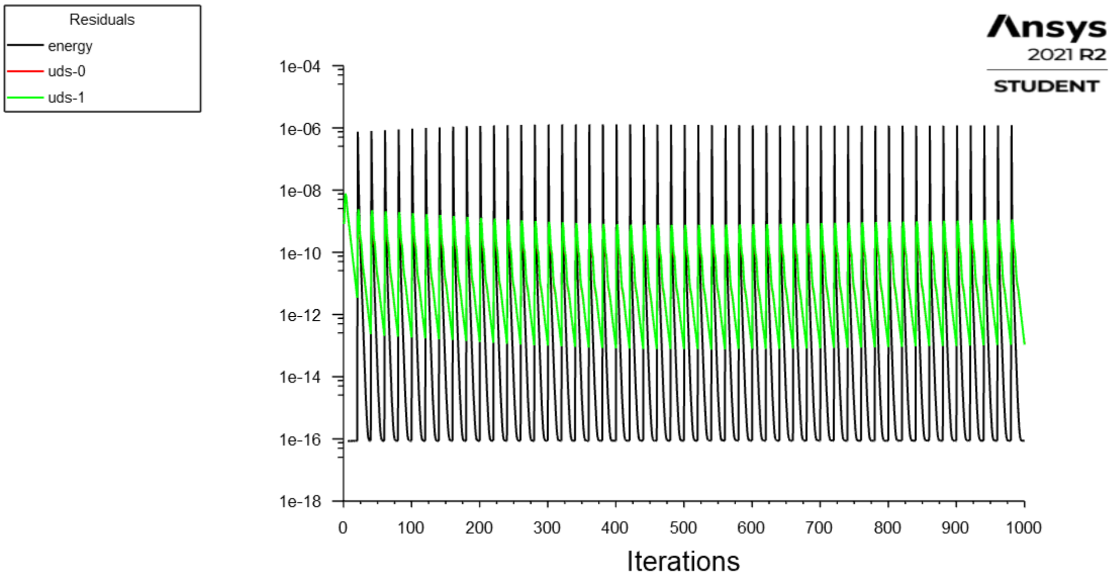

Fig. 4.5 Scaled residuals plot (without cooling)

The residuals and report plots obtained are similar to the ones in the single battery cell, but with an expected increase in magnitude.

Post-processing results (without cooling condition)

The same process is repeated as in post-processing done for single-cell (without cooling condition) and contours are shown in the below figures.

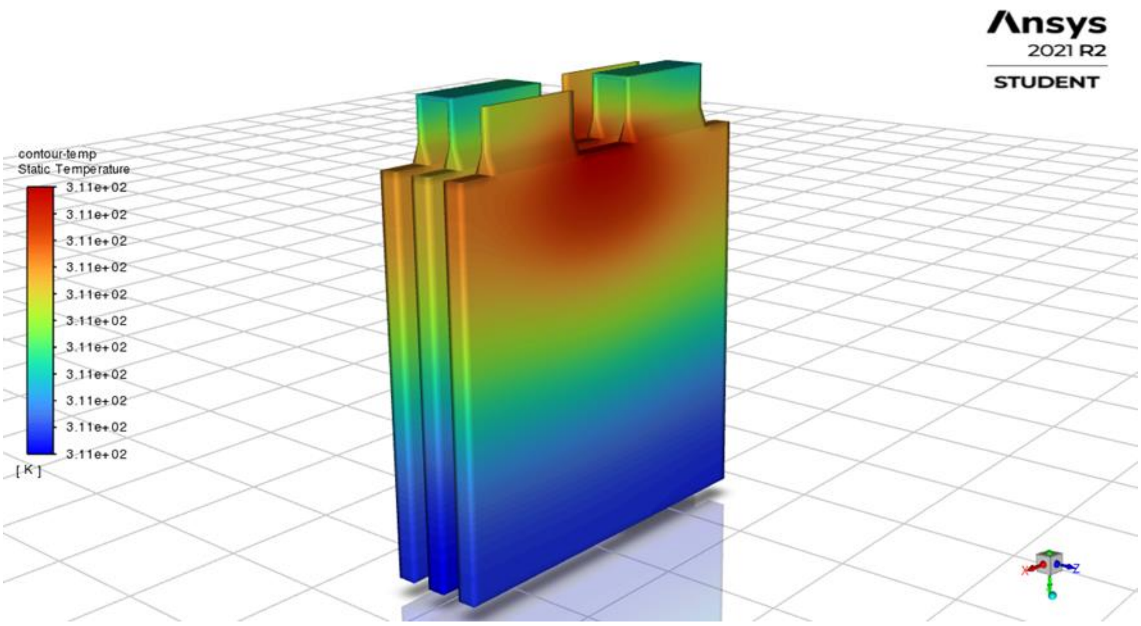

Fig. 4.6 Static temperature contour (without cooling)

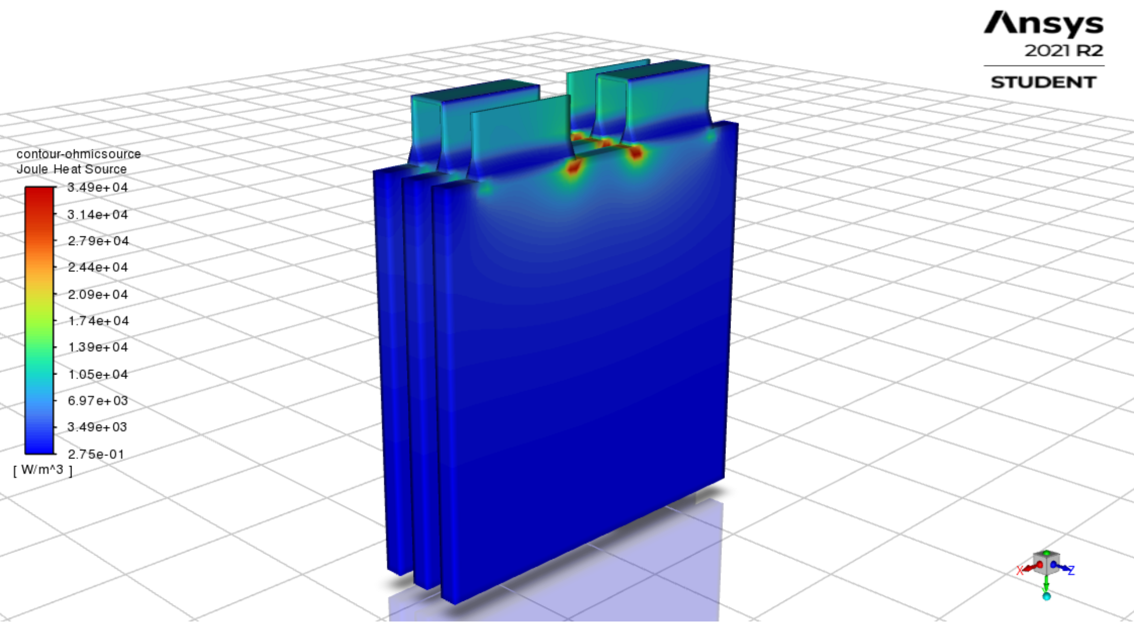

Fig. 4.7 Joule heat source contour (without cooling)

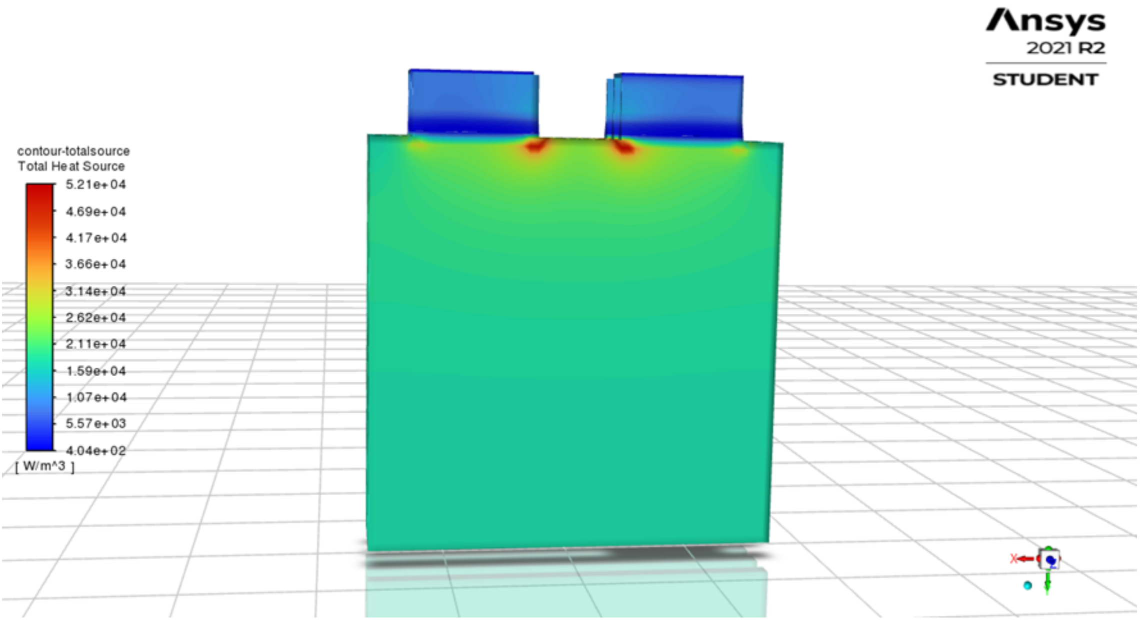

Fig. 4.8 Total heat source contour (without cooling)

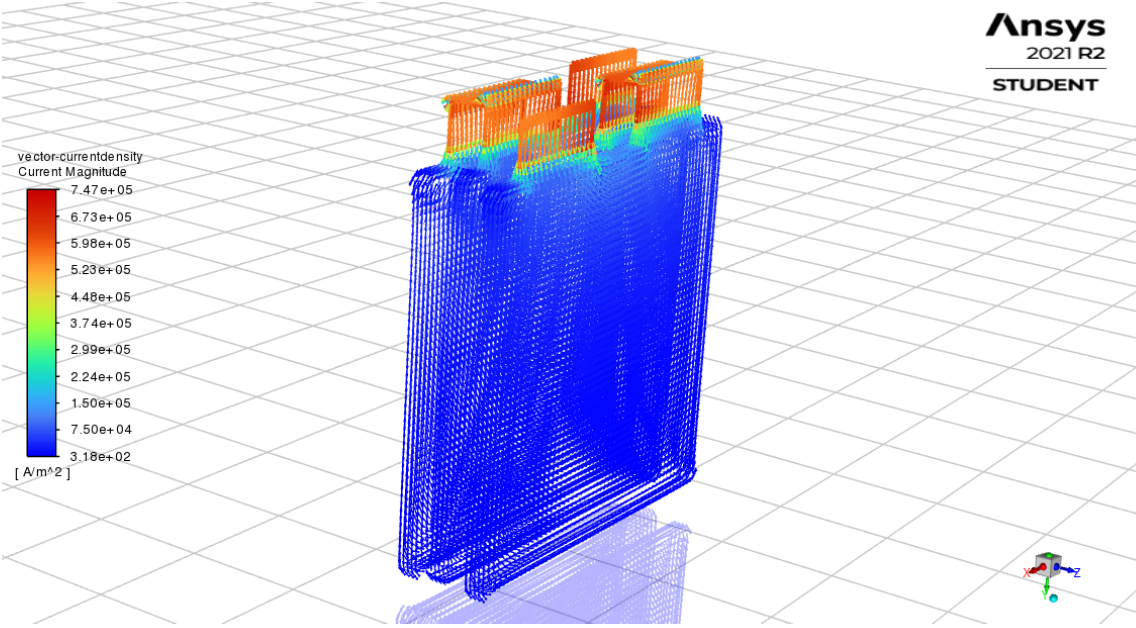

Fig. 4.9 Current density vectors (without cooling condition)

B) SIMULATING A 1P3S BATTERY PACK USING THE BATTERY MODEL (with cooling)

The change in procedure occurs only in the case of assigning boundary conditions. For "without cooling" model, there is no consideration of any heat transfer (h=0), implying no cooling taking place. Here, in "with cooling" model, forced heat convection is taken (h=20), implying the transfer of heat to external materials.

4.4 Solutions and graphs (with cooling condition)

The process used for single-cell with forced convection is repeated for the pack as well and the results are obtained as shown in the figures below.

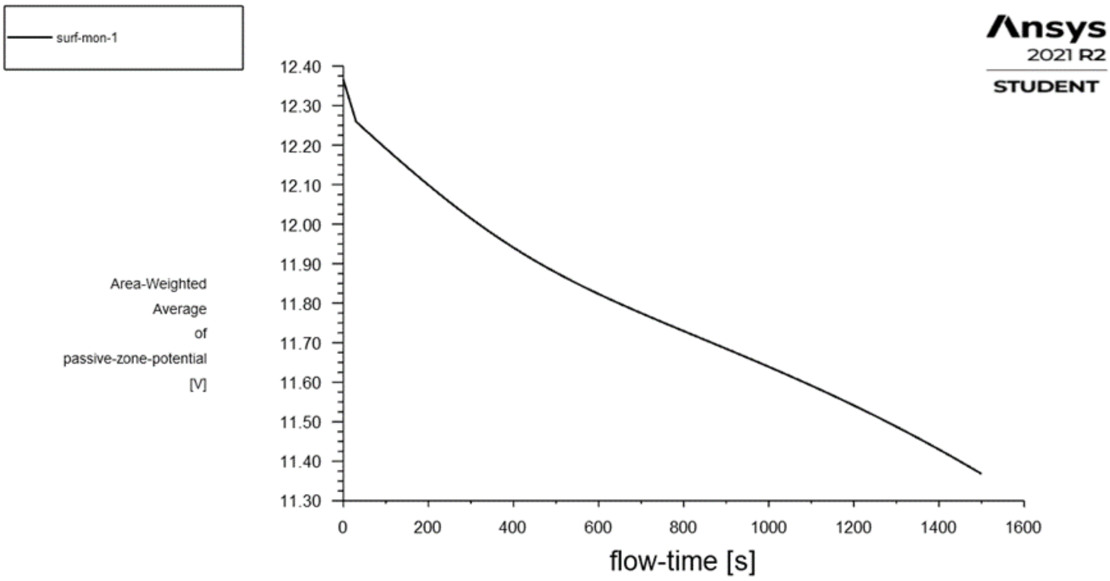

Fig. 4.10 Passive zone potential plot (with cooling)

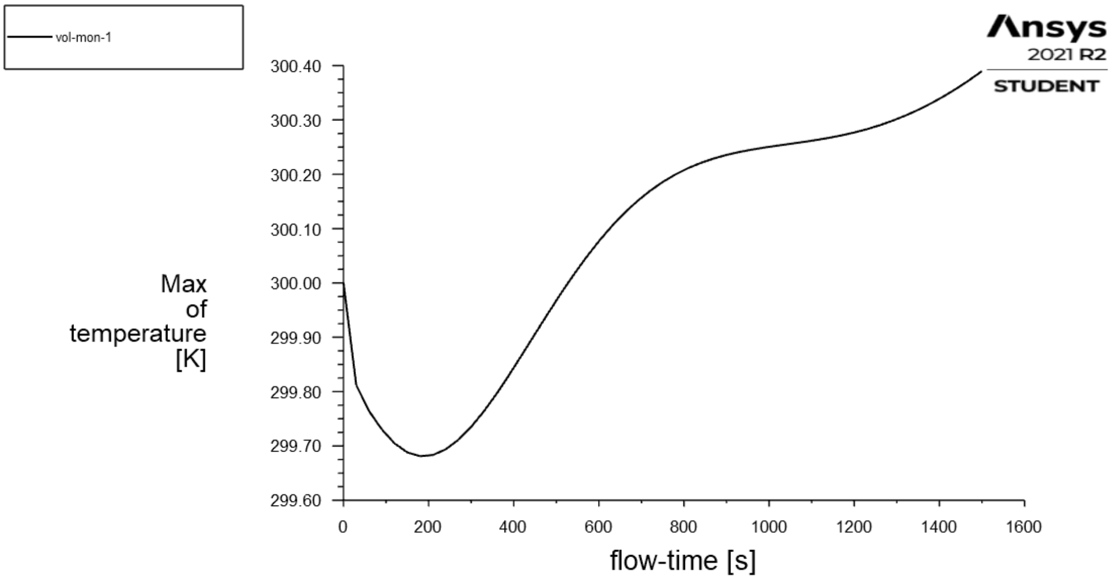

Fig. 4.11 Max Temperature plot (with cooling)

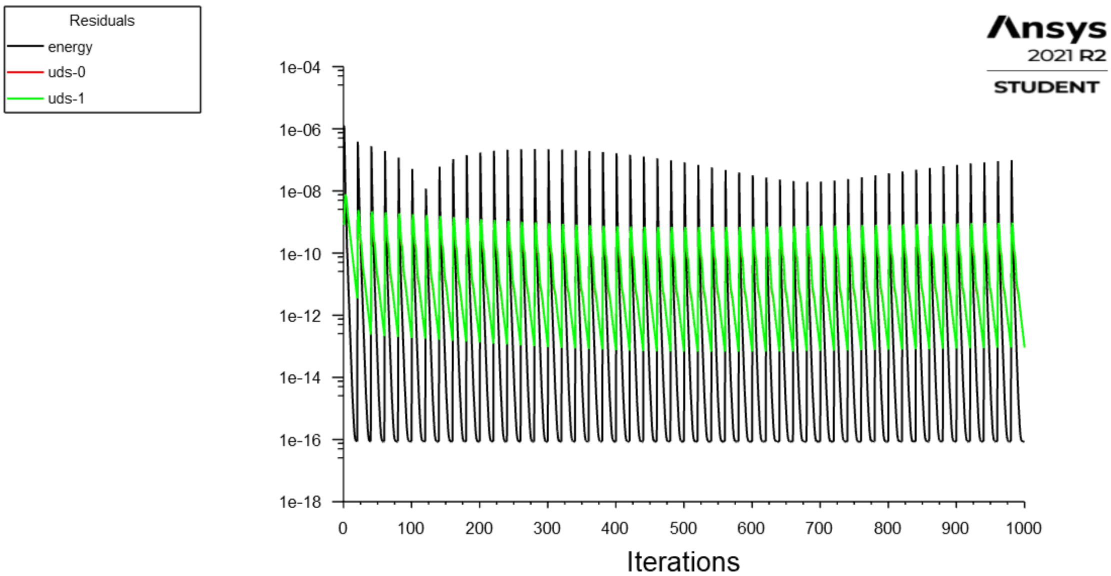

Fig. 4.12 Residuals plot (with cooling)

The residuals and report plots similar to the ones in the single battery cell, but with an expected increase in magnitude are obtained.

Post-processing results (with cooling condition)

The same process is repeated as in post-processing done for single-cell (with cooling condition) and contours are shown in the below figures.

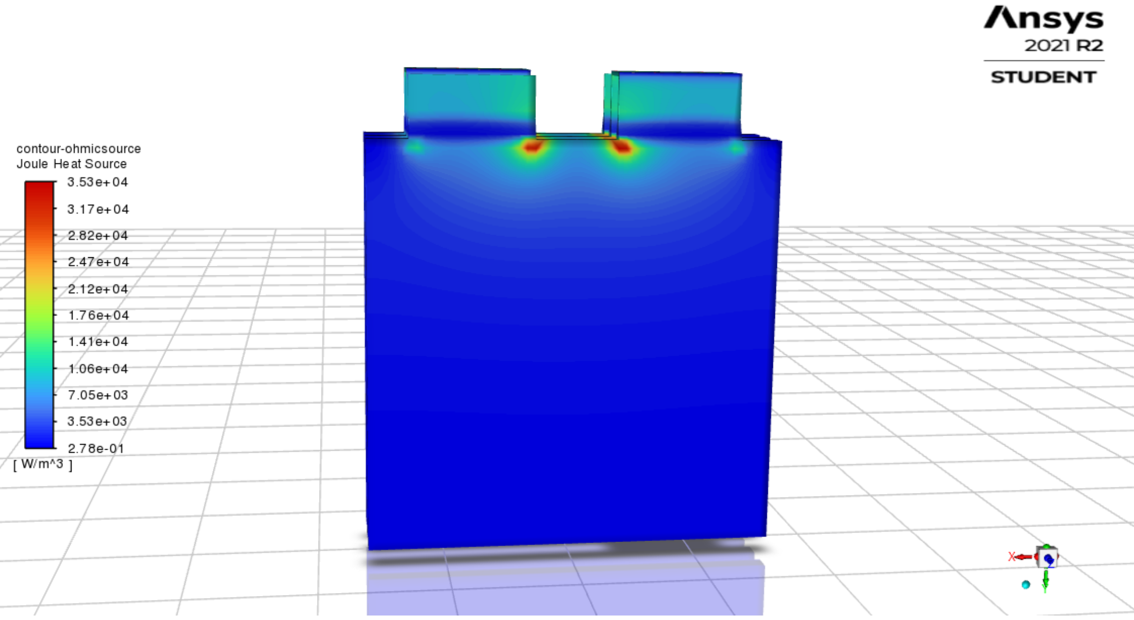

Fig. 4.13 Joule heat source contour (with cooling)

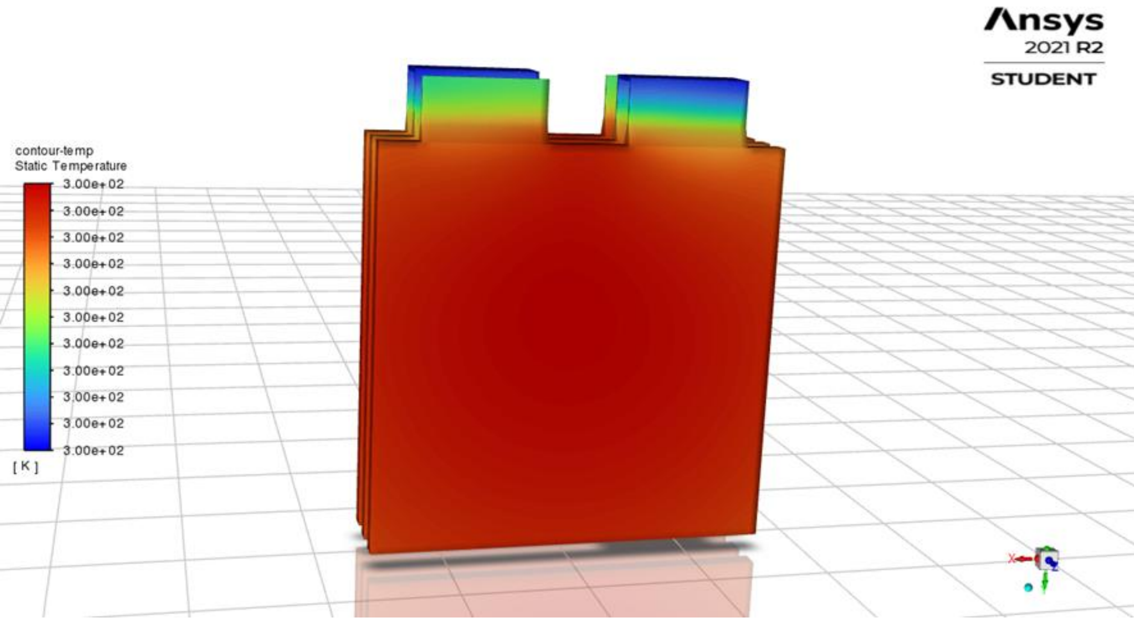

Fig. 4.14 Static temperature contour (with cooling) shows the variation of temperature along the cell body.

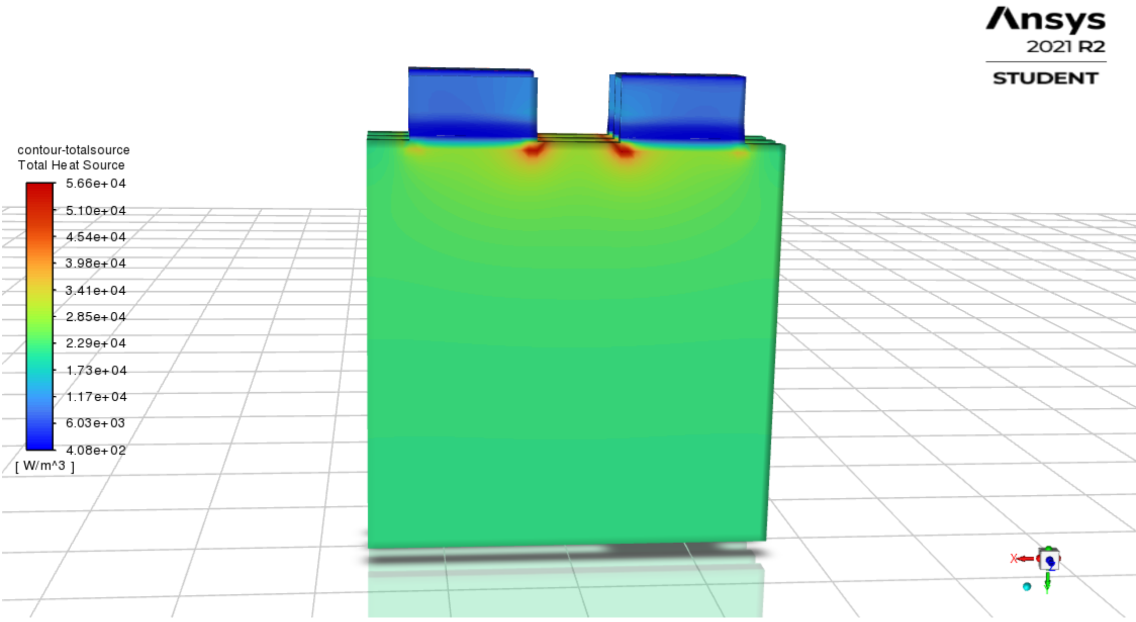

Fig. 4.15 Total heat source contour (with cooling)

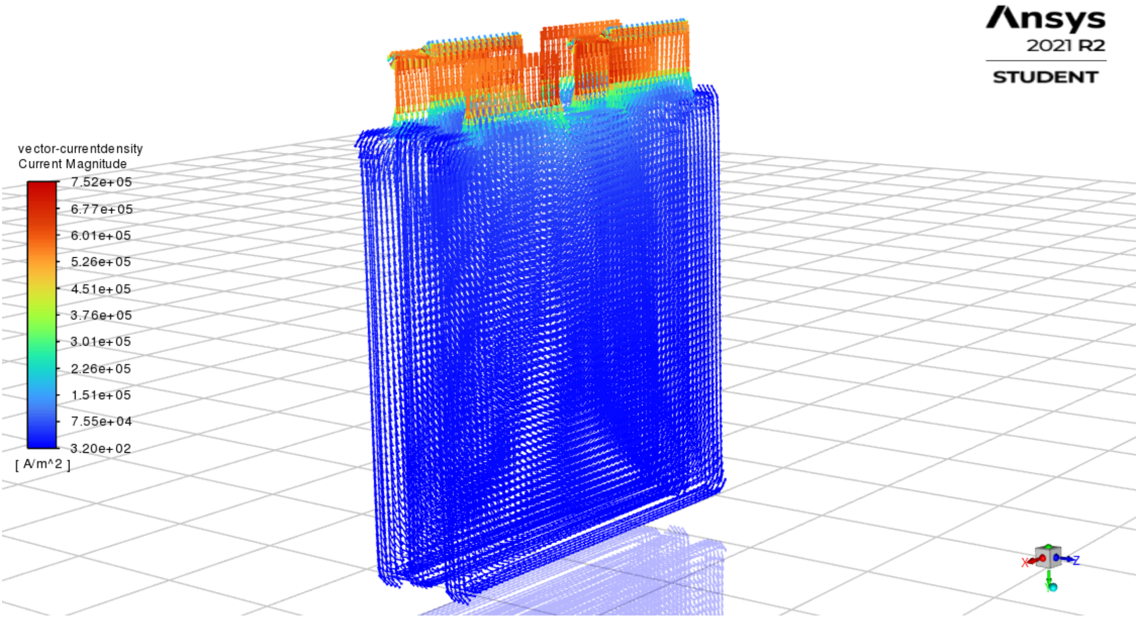

Fig. 4.16 Current density vectors (with cooling condition)

CHAPTER – 5 COMPARISION OF RESULTS

The following table is drawn as a comparison chart, denoting all the obtained values of parameters considered during the analysis of the single-cell battery and the battery pack.

Parameters	Cell (h=0)	Cell (h=20)	Pack (h=0)	Pack (h=20)
Total heat source avg. (W/m^3)	-	-	2.52e+4	2.85e+4
Joule heat source avg. (W/m^3)	-	-	1.63e+4	1.73e+4
Anode Potential avg. (V)	-1.91e+3	-1.89e+3	-	-
Cathode Potential avg. (V)	3.49	3.48	-	-
Static temperature (K)	307.5	298.5	311	300
Current density-max (A/m^2)	8.23e+4	8.52e+4	7.3e+5	7.52e+5
Max temperature (K)	308	298.6	312	300.4
Passive zone potential after the last timestep (V)	3.5	3.49	11.8	11.4

TABLE 4.3 Comparison of results.

CHAPTER – 6 CONCLUSION

Thus, CFD analysis of the considered single-cell battery and battery pack is done. The anode potentials for single-cell battery before and after cooling are -1.19e+3 and -1.89e+3 volts respectively. The current densities for single cell battery, single cell (with cooling), battery pack and battery pack (with cooling) are found to be 8.23e+4, 8.52e+4, 7.3e+5 and 7.52e+5 A/m^2 respectively. The results show that cooling of battery produces decreases temperature and increases current density, thereby, it is necessary to prevent battery degradation or damage from various thermal thresholds. Moreover, this reduces the overall power consumption of the system. This increases the battery life and minimizes losses in useful energy.

REFERENCES

1. U. S. Kim et al, "Effect of electrode configuration on the thermal behaviour of a lithium polymer battery", Journal of Power Sources, Volume 180 (2), pages 909-916, 2008.

2. U. S. Kim, et al., "Modeling the Dependence of the Discharge Behaviour of a Lithium-Ion Battery on the Environmental Temperature", J. of Electrochemical Soc., Volume 158 (5), pages A611-A618, 2011.

3. ANSYS official fluent module guide 2021

4. US OEER. A Grand Challenge in Plug-In Electric Vehicles, Office of Energy Efficiency & Renewable Energy.