

**EXPERIMENTAL AND NUMERICAL INVESTIGATION OF
BATTERY COOLING USING ETHYLENE GLYCOL
THROUGH CORRUGATED CHANNEL**

PROJECT REPORT

Submitted by

NIHALA AMEEN

TKM20MEIR10

to

*APJ Abdul Kalam Technological University
in partial fulfillment of the requirements for the award of
Master of Technology in Mechanical Engineering.
(Industrial Refrigeration and Cryogenic Engineering)*



Department of Mechanical Engineering

TKM College of Engineering, Kollam

September 2022

DEPARTMENT OF MECHANICAL ENGINEERING
TKM COLLEGE OF ENGINEERING, KOLLAM



CERTIFICATE

Certified that this report entitled '*Experimental and numerical investigation of battery cooling using ethylene glycol through corrugated channel*' is the report of project presented by **NIHALA AMEEN, TKM20MEIR10** during **2021-2022** in partial fulfilment of the requirements for the award of the Degree of Master of Technology in Mechanical Engineering (Industrial Refrigeration and Cryogenic Engineering) of APJ Abdul Kalam Technological University.

Guide:

Dr. Vishnu S.B.
Assistant Professor
Dept. of Mechanical Engineering
TKM College of Engineering, Kollam

Coordinator:

Dr. Shafi K A

Professor
Dept. of Mechanical Engineering
TKM College of Engineering, Kollam

Dr. Dilcep P N

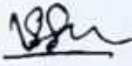
Head of the Department
Dept. of Mechanical Engineering
TKM College of Engineering, Kollam

DECLARATION


I, NIHALA AMEEN hereby declare that, this project report entitled **experimental and numerical investigation of battery cooling using ethylene glycol through corrugated channel** is the bonafide work of mine carried out under the supervision of Dr. Vishnu S.B, Professor, Dept. of Mechanical Engineering TKM College of Engineering, Kollam. I declare that, to the best of my knowledge, the work reported herein does not form part of any other project report or dissertation on the basis of which a degree or award was conferred on an earlier occasion to any other candidate. The content of this report is not being presented by any other student to this or any other University for the award of a degree.

NIHALA AMEEN

University Register No: TKM20MEIR10 of year 2020-2022

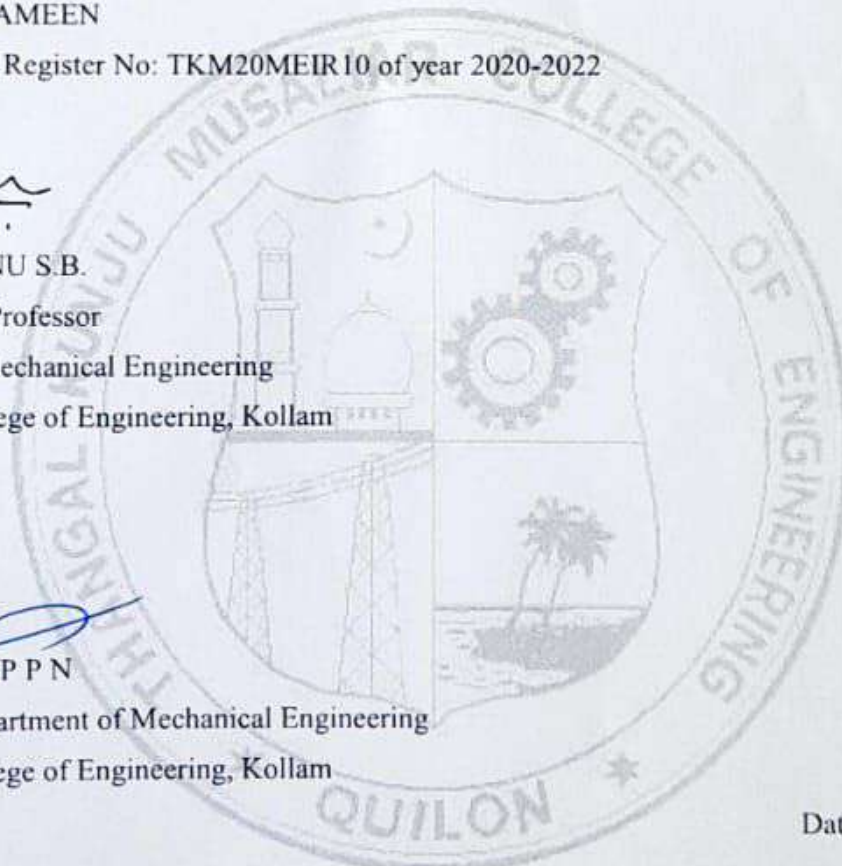


Dr. VISHNU S.B.
Assistant Professor
Dept. of Mechanical Engineering
TKM College of Engineering, Kollam



Dr. DILEEP P N
Head, Department of Mechanical Engineering
TKM College of Engineering, Kollam

Date: 31/08/2022



ACKNOWLEDGEMENTS

I take this opportunity to express my deep sense of gratitude and sincere thanks to all who helped us to complete the seminar successfully.

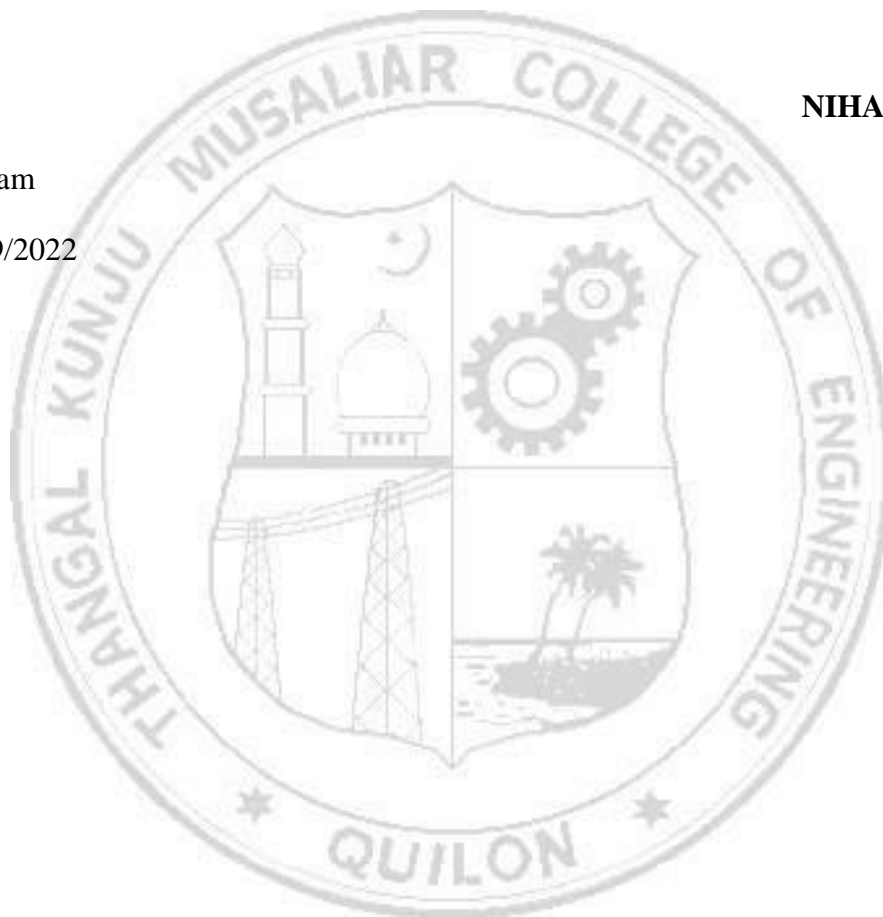
I am deeply indebted to my guide **Dr. Vishnu S.B**, Professor, Department of Mechanical Engineering for her excellent guidance, positive criticism and valuable comments. I am greatly thankful to **Dr. Dileep P N**, Head of Mechanical Engineering Department for his support and cooperation.

Finally, I thank our parents and friends near and dear ones who directly and indirectly contributed to the successful completion of my project.

NIHALA AMEEN

Place: Kollam

Date: 12/09/2022



ABSTRACT

As the automotive industry progresses, electric vehicles (EV) grow with increasing demand throughout the world. Nickel metal hydride (NiMH) battery and lithium-ion (Li-ion) are widely used in EV due to their advantages such as impressive energy density, good power density, and low self-discharge. However, the batteries must be operated within their optimum range for safety and good thermal management to enable a longer lifespan, lower costs, and improve safety for EV batteries. But most of the cases such as fast charging, battery discharge it is not possible to keep the designed operating conditions of battery. So, battery thermal management system is necessary for good operating conditions of the battery. Most of the cases liquid cooling method is used with coolant as water. But water can't be used in all countries because of the freezing point of the water. So, ethylene glycol and water mixture is proposed. In this work experimental study of different coolants such as water, ethylene glycol, ethylene glycol and water mixture in the ratio of 40:60, 50:50, 60:40 and numerical simulation of water, ethylene glycol and ethylene glycol - water in the ratio of 60:40 is carried out. From the comparison between different coolants it can be observed that ethylene glycol and water with ratio of 60:40 was more effective than others.

Keywords: Liquid cooling, ethylene glycol and water, lithium ion battery, Battery thermal management system.

CONTENTS

Title	Page Number
List of Figures	<i>vi</i>
List of Tables	<i>viii</i>
Chapter-1. Introduction	1
1.1 Battery pack	3
1.2 Thermal runaway	
Chapter-2. Literature Review	6
2.1 Objectives	10
2.2 Methodology	10
Chapter-3. Battery thermal management systems	11
3.1 Battery Thermal Management Systems	11
3.2 Air cooling	11
3.3 Liquid cooling	12
3.3.1 Indirect Cooling system	13
3.3.2 Tab Cooling	14
3.3.3 Immersion Cooling	15
3.3.4 Direct Refrigerant Cooling	16
3.3.5 Phase Change Material	16
3.3.6 Thermoelectric Cooling	19
3.3.7 Heat Pipe Cooling	20
Chapter-4. Experimental method	22
4.1 Experimental Setup	22
4.2 Experimental Procedure	25

4.3 Experiments Using Water In Corrugater Channel	25
4.4 Experiments Using Ethylene Glycol In Corrugater Channel	26
4.5 Experiments Using Ethylene Glycol And Water In Corrugater Channel In The Ratio 40:60	27
4.6 Experiments Using Ethylene Glycol And Water In Corrugater Channel In The Ratio 50:50	27
4.7 Experiments Using Ethylene Glycol And Water In Corrugater Channel In The Ratio 60:40	28
Chapter-5. Applications Of Cfd	30
5.1 Aerospace / Mechanical Engineering	30
5.1.1 Aerospace / Mechanical Engineering	31
5.1.2 Ocean And Marine Engineering	31
5.1.3 Environmental Engineering	31
5.1.4 Electrical and Electronic Engineering	31
5.1.5 Chemical process Engineering	31
5.1.6 Biomedical Engineering	31
5.2 Steps Involved In Performing Cfd Simulation Of A Flow Problem	31
5.2.1 Finite Difference Method (FDM)	32
5.2.2 Finite Element Method (FEM)	32
5.2.3 Finite Volume Method (FVM)	32
5.3 Working Of A Cfd Code	33
5.3.1 Pre-processor	33
5.3.2 Solver	34
5.3.3 Post-processor	34
5.4 Computational Fluid Dynamics Simulation	34
5.5 Phases Of Modelling And Simulation	36
5.6 CFD Calculation	37

Chapter-6. Computational Domain	39
6.1 Steady State Analysis	39
6.2 Modelling	39
6.3 Meshing	41
6.4 Fluent Analysis	42
6.4.1 Boundary Conditions	42
6.4.2 Input Parameters	42
6.4.3 Convergence Criteria	43
Chapter-7. Results And Discussion	44
7.1 Mesh Independent Study	45
7.2 Validation Of Numerical Results	45
7.3 Selection Of The Best Coolant	49
7.4 Comparison Between Experimental Data Numerical Date	50
Chapter-8. Conclusion	51
References	52

LIST OF FIGURES

Title	Page Number
Fig. 1.1 Charging and Discharging Phase of Lithium-ion battery.	2
Fig. 3.1 Air cooling method.	11
Fig. 3.2 Indirect Liquid Cooling	13
Fig. 3.3 Immersion cooling	16
Fig. 3.4 Phase change material	17
Fig. 3.5 Thermoelectric cooling	20
Fig. 4.1 Corrugated channel	22
Fig. 4.2 Heat exchanger used for experiment	23
Fig. 4.3 Pump used for experiment	23
Fig. 4.4 Experimental setup	24
Fig. 4.5 Schematic diagram of experimental setup	24
Fig. 4.6 Experiments using water in corrugated channel	26
Fig. 4.7 Experiments using EG in corrugated channel	26
Fig. 4.8 Experiments using EG-water (40:60) in corrugated channel	26
Fig. 4.9 Experiments using EG-water (50:50) in corrugated channel	28
Fig. 4.10 Experiments using EG-water (60:40) in corrugated channel	29
Fig. 6.1 The dimensions of BTMS	36
Fig. 6.2 Numerical computational domain	40
Fig. 6.3 Mesh domain	41
Fig. 6.4 Fluid domain	41
Fig. 6.5 Heater domain	41
Fig. 7.1 Variation of average temperature with number of elements	45

Fig. 7.2 Maximum surface temperature of corrugated channel using water	46
Fig. 7.3 Maximum surface temperature of corrugated channel using EG	46
Fig. 7.4 Maximum countour of corrugated channel using EG	47
Fig. 7.5 Velocity vector of corrugated channel using EG	47
Fig. 7.6 Maximum surface temperature of corrugated channel using EG-water (60:40)	48
Fig. 7.7 Temperature contour of corrugated channel using EG-water (60:40)	48
Fig. 7.8 Velocity vector of corrugated channel using EG-water (60:40)	49
Fig. 7.9 Comparison between different coolant performance	49
Fig. 7.10 Comparison between experimental data vs numerical data	50

LIST OF TABLES

Title	Page Number
Table 6.1 Dimensions of Computational domain	40
Table 6.2 Input parameters	42
Table 7.1 Material properties used in numerical solutions	44

CHAPTER 1

INTRODUCTION

Lithium-ion batteries (LIBs) have been rapidly adopted for uses ranging from consumer electronics to electric vehicles (EVs) and even grid-scale balancing of renewable electricity output. However, improvements in: cost, energy density, power density, lifetime, safety, operating temperature, predictability and recyclability are still needed.

Effective battery thermal management systems (BTMSs) are now considered essential to maximise lifetime, enhance power capability and reduce the risk of thermal runaway. The aim of these systems is to remove heat from a battery pack, thus regulating the operating temperature, and to homogenise temperature within individual cells and between different cells of a pack. Effective battery thermal management systems (BTMSs) are now considered essential to maximise lifetime, enhance power capability and reduce the risk of thermal runaway. The aim of these systems is to remove heat from a battery pack, thus regulating the operating temperature, and to homogenise temperature within individual cells and between different cells of a pack.

Many BTMSs currently exist ranging from passive air cooling to indirect liquid-based solutions using cooling plates. Liquid based systems are generally able to buffer and remove a larger amount of heat than air-cooled systems, due to their superior convective heat transfer coefficient and specific heat capacity. However, this additional performance often comes at the cost of increased complexity and system weight. Many BTMSs currently exist ranging from passive air cooling to indirect liquid-based methods using cooling plates. Liquid based systems are often able to buffer and remove a larger quantity of heat than air-cooled systems, due to their superior convective heat transfer coefficient and specific heat capacity. However, this improved performance frequently comes at the expense of a heavier system and increased complexity.

1.1 BATTERY PACK

In the periodic table's alkaline group, lithium is a very light metal. It has three electrons and an electronic configuration of $1s^2, 2s^1$. Lithium has the highest tendency to lose an electron, and this property makes lithium highly unstable. Lithium is more stable in its form as lithium metal oxides. Due to the extremely high reactivity of the metal, lithium-ion cells can achieve very high voltages on their own. A lithium-ion

battery is made up of a number of modules connected in series, each of which is made up of separate cells connected in parallel and series. A lithium-ion battery has three primary parts: Lithium Metal oxide, Electrolyte, Graphite. Graphite and lithium metal oxide are separated by an electrolyte.

Charging and discharging are the two phases of how lithium-ion batteries operate. It joins the cell to a power source during the charging phase. Graphite is connected to the negative terminal (cathode), and lithium metal oxide is connected to the positive terminal (anode) (cathode). Lithium's valence shell contains an electron that is drawn to the positive terminal of the power source. Electrolyte serves as a barrier that prevents electrons from passing. Lithium-ion (Li^+) travels through the electrolyte and becomes trapped in the area between the graphite while electrons travel through the external supply and reach the graphite layer. The cell is fully charged when all of the lithium-ion is squeezed between the graphite solid sheet.

Because lithium-ion and electron are highly unstable when created during the charging stage, the battery begins to discharge when the power source is replaced by a load. To create a more stable lithium metal oxide state, the lithium-ion flows toward the metal oxide through the electrolyte. Through the load, the electrons begin to move in the direction of the anode, and we thereby obtain electric current. The battery discharges when all the electrons return to a more stable condition and the lithium-ion material. The graphite used in the cell serves as a container for lithium ions rather than actively participating in the chemical reaction. Along with electricity, this process also generates heat.

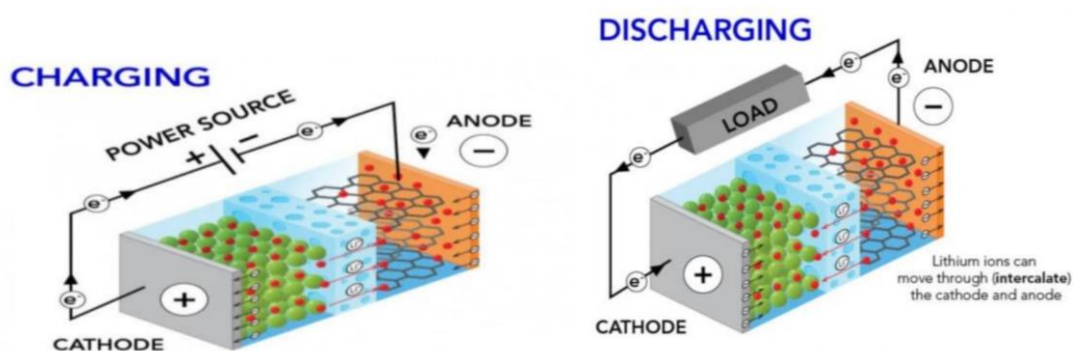


Fig 1.1 Charging and Discharging Phase of Lithium-ion battery (Chaitanya Pandya et al, 2021)

Lithium-ion batteries produce heat as a result of their intricate internal workings, including exothermic chemical reactions, ohmic resistance, and cell polarisation brought on by differences in the battery potential during open circuit charging and discharging. Thermal runaway occurs in lithium-ion batteries under specific circumstances. The process known as thermal runaway occurs when the materials inside these batteries begin to break down exothermically and a lot of heat is produced.

This is a runaway reaction because it cannot be stopped when the heat cannot be expelled as quickly as it is produced. In this case, the cell bursts if the appropriate cooling technique is not applied to remove the heat created. A cell explosion has negative effects. Choosing the appropriate cooling technique for such a battery pack is crucial. We concentrate on several cooling techniques in this research.

1.2 THERMAL RUNWAY

One of the main requirements for BTMSs is to prevent thermal runaway (TR). Abuse involving mechanical, electrical, and/or thermal components may be the primary cause of TR. The most common cause of mechanical abuse is mechanical deformation of the battery, which happens during car collisions, followed by extrusion and puncture, which causes the separator to break and potentially short the cell. Electrical abuse is typically brought on by external short circuits, excessive battery charging or discharging, which promotes dendrite growth and separator penetration.

Thermal abuse is typically brought on by discrepancies between a single cell's internal resistance and heat dissipation, as well as an abnormal increase in contact resistance, which results in the separator collapsing on a broad scale. Internal short circuit serves as a connecting mechanism in this case for the processes that start TR. Cell deformation brought on by mechanical abuse could result in an internal short circuit and electrical abuse.

Thermal abuse results from the combination of this electrical abuse with joule heating and chemical reaction heat. Thermal abuse then causes the temperature to rise, which starts the TR chain reaction. TR in one cell can spread to adjacent cells via convection, radiation, and conduction, which results in TR for the entire pack.

The total heat released (THR), which is generated by integrating the heat release rate (HRR) curves, is a crucial measure for assessing the fire risk of LIBs. The specific energy, capacity, and SOC of the battery all affect the HRR and THR's size. Batteries with 100% SOC have the highest THR based on the HRR and THR. Additionally, the HRR and THR of LIB are directly correlated with the external environmental pressure; the lower the environmental pressure, the lower the HRR and THR of LIB.

This is mostly due to the fact that as pressure drops, the absolute oxygen concentration at the battery surface also does so. This results in a slower combustion reaction rate and less heat feedback from the flame. Low pressure also lessens the thermal convection effect of flame on lithium batteries, which has an impact on the HRR and THR. The SOC and battery chemistry must be taken into consideration when determining if a BTMS can stop TR.

Venting takes place during a thermal runaway. Due to the extraordinarily high internal pressures created, venting frequently occurs during TR of LIBs. Diverse substances, including gases, liquids, and solids, are ejected from the cells during the venting process. Carbon dioxide (CO₂), carbon monoxide (CO), hydrogen (H₂), and short-chain hydrocarbons, such methane (CH₄) and ethylene (C₂H₄), make up the majority of the gaseous ejection products, the majority of which are combustible.

SOC, cathode material, and electrolyte are only a few of the cell's many characteristics that affect the composition, concentration, and volume of gaseous ejection products. The presence of fluoride gases like phosphoryl fluoride (PF₃) and hydrogen fluoride (HF) (POF₃). They may be in little quantities, yet they can still be hazardous. Zhang et al. found water and electrolytes such as ethyl methyl carbonate (EMC), diethyl carbonate (DEC), dimethyl carbonate (DMC), and ethylene carbonate in the liquid ejection products (EC). These organic vapours pose a risk to people's health.

A list of the different liquid and gaseous ejection products that were discovered using a combination of gas chromatography, mass spectrometry, and ion chromatography. Additionally, because of their high temperature, toxicity, and environmental contamination, solid ejection products can also be risky. In order of highest to lowest content, the elemental compositions contain carbon (C), nickel (Ni), copper (Cu), cobalt (Co), manganese (Mn), aluminium (Al), and lithium (Li). It is therefore essential to be aware of the different materials that are expelled from a cell in

order to create effective BTMSs, particularly in the case of combustible components that may require non-flammable cooling fluids.

CHAPTER 2

LITERATURE REVIEW

Electric vehicles (EV) are in higher demand globally as the automotive industry develops. Due to their benefits including outstanding energy density, good power density, and minimal self-discharge, lithium-ion (Li-ion) and nickel-metal hydride (NiMH) batteries are frequently utilised in electric vehicles. To enable a longer lifespan, reduced cost, and better safety for EV batteries, the batteries must be operated within their optimal range for safety and good thermal management. As a result, rather of relying solely on conventional cooling systems, the requirement for a liquid cold plate (LCP) to be employed in EV batteries is now heavily dependent on the distribution of the desired temperature. The LCP's fin configuration would also affect how effectively the EV battery is cooled.

Abdullah Mansur Aldosry and colleagues identified the improvement in heat transfer of liquid cold plate systems with the oblique fin and various liquid coolants. G13 ethylene glycol and distilled water are the two liquid types used in the experiment. The liquid concentrations are 10% ethylene glycol, 100% distilled water, 75% ethylene glycol plus 25% distilled water, 50% ethylene glycol plus 50% distilled water, and 25% ethylene glycol plus 75% distilled water. To increase the efficiency of flowing fluid and heat transmission with the gate door valve, three alternative flow rates—0.3, 0.5, and 0.7 GPM—have been used.

In this experiment, they looked at how employing liquid coolants and oblique fins inline configurations could speed up heat transmission. The experiment's findings demonstrated that the oblique fins considerably improved the performance of the liquid cold plate and that the flow rate enhanced the performance of heat transfer. The battery surface temperature was kept below 5°C, which is the safe operating temperature for an electric vehicle battery, at a constant temperature of 100°C, 75% of distilled water, and 25% of ethylene glycol at a flow rate of 0.7 GPM.

A dual-purpose cooling plate for prismatic lithium-ion batteries (LIBs) to increase the battery pack life and safety for applications in vehicles, aircraft, and stationary electric storage systems for grid and renewables is proposed. Abdul Haq

mohammed et.al proposed cooling plate it can maintain the temperature of the battery within the manufacturers' recommended temperatures during the battery normal operation to increase the battery life, and can effectively control and dissipate the generated heat during thermal runaway to prevent propagation of heat from one cell to the adjacent cells in the battery pack to increase the safety. The proposed cooling plate is made of aluminium and works on the principle of the liquid cooling method with 60% ethylene glycol in water as the coolant. The cooling plate is featured with pins with staggered arrangement, which act as heat sinks and disperse the coolant over the surface of the cooling plate. The performance of the cooling plate is examined for heat generation trend of aggressive discharging of the battery during the normal operation and the heat generation during thermal runaway of the fully charged battery. The results show that the designed cooling plate can maintain the LIB surface temperature below 25 °C with the input coolant temperature of 20 °C and flow rate of 0.2 L/min during the normal operation, and can control the LIB temperature during thermal runaway at about 75 °C in 30 s with the input coolant temperature of 20 °C and flow rate of 30 L/min. The pressure-drop of the coolant is also about 75 Pa for the normal operation mode and about 54 kPa for the thermal runaway operation mode of the cooling plate.

Dattatraya G. Subhedar et.al, studies the heat transfer potential of Al₂O₃/Water-Mono Ethylene Glycol nanofluids is investigated experimentally as a coolant for car radiators. The base fluid was the mixture of water and mono ethylene glycol with 50:50 proportions by volume. The stable nanofluids obtained by ultra-sonication are used in all experiments. In this study nanoparticle volume fraction, coolant flow rate, inlet temperature used in the ranges of 0.2–0.8%, 4–9 l per minute and 65–85 °C. The results show that the heat transfer performance of radiator is enhanced by using nanofluids compared to conventional coolant. Nanofluid with lowest 0.2% volume fraction 30% rise in heat transfer is observed. Also, the estimation of reduction in frontal area of radiator if base fluid is replaced by Nanofluid is done which will make lighter cooling system, produce less drag and save the fuel cost.

Battery thermal management systems are critical for high performance electric vehicles, where the ability to remove heat and homogenise temperature distributions in single cells and packs are key considerations. Immersion cooling, which submerges the battery in a dielectric fluid, has the potential of increasing the rate of heat transfer by 10,000 times relative to passive air cooling. In 2-phase systems, this performance

increase is achieved through the latent heat of evaporation of the liquid-to-gas phase transition and the resulting turbulent 2-phase fluid flow. However, 2-phase systems require additional system complexity, and single-phase direct contact immersion cooling can still offer up to 1,000 times improvements in heat transfer over air cooled systems. Fluids which have been considered include: hydrofluoroethers, mineral oils, esters and water-glycol mixtures. In this review Carlotte Roe et.al, presents the current state-of-the-art in immersion cooling of lithium-ion batteries, discussing the performance implications of immersion cooling but also identifying gaps in the literature which include a lack of studies considering the lifetime, fluid stability, material compatibility, understanding around sustainability and use of immersion for battery safety. Insights from this review will therefore help researchers and developers, from academia and industry, towards creating higher power, safer and more durable electric vehicle.

The temperature rise is the major factor that influences the functioning of Lithium-ion batteries (Li-Ion). To refine the heat efficiency of the battery there are various methods to dissipate the heat. Selecting a correct cooling technique for a Li-ion battery module of an electric vehicle (EVs) and deciding an ideal cooling control approach to maintain the temperature between 5° C to 45° C is necessary. Maintaining an optimal temperature is essential as it increases safety, reduces maintenance cost, and increases the service life of the battery pack. When choosing a cooling technique various trade-offs are made among various parameters like weight, cooling effect, temperature consistency, and cost. In this paper four lithium-ion battery cooling methods: liquid cooling, phase changing material cooling, dielectric oil cooling, and thermoelectric cooling is discussed. Chaitanya Pandya reviewed an elaborate study on Advantages, Disadvantages, and Applications of these four types of cooling systems.

The mixture of water with either ethylene or propylene-glycol is necessary to lower its freezing point and eliminate ice formation. However, the boiling point of deionized water can be pushed up by mixing it with anti-freezing materials like ethylene glycol (EG). Hence R. Prasanna Shankara present research work focuses on the heat transfer characteristics of nano-fluids with different combinations of EG and deionized water (60:40, 30:70, and 20:80) along with graphene oxide (GO) nanoparticles (0.1 % wt.). The heat transfer potential of graphene oxide/ deionized water-ethylene glycol nanofluids is experimentally investigated in coolant application for car radiators, in

which the coolant flow rate is varied from 180 to 420 (LPH) at a fixed coolant temperature of 90°C. The evaluation of stability is performed using visual inspection and zeta potential test. The density of nanofluid was determined by oscillating U-Tube density meter. The optimized combination of 60% EG, 40% DW and 0.1 wt% GO exhibited higher heat transfer characteristics of the radiator system used. The maximum heat transfer enhancement of 42.77% at 300 LPH, 18.14% at 360 LPH, and 71.1% at 240 LPH for 60:40 EG/DW-based GO nanofluids were obtained. The maximum Nusselt number value of 192 at 420 LPH was observed for 60:40 EG/DW-based GO nanofluid as compared with 20:80, 30:70 EG/DW-based GO nanofluid. It is estimated that by replacing conventional coolant with this nanofluid, reduction in the frontal area of radiator is done.

The cooling capabilities of an ethylene glycol (EG)-based nanofluid containing three different types of nanoparticles: copper oxide (CuO), aluminium oxide (Al₂O₃), and titanium dioxide (TiO₂) are investigated by Winifred Nduku Mutuku. Nanofluids have enhanced thermophysical properties, hence they can be used in a plethora mechanical and engineering applications such as nanofluid coolant: electronics cooling, vehicle cooling, transformer cooling, computers cooling and electronic devices cooling. A model depicting the vertical fluid flow in a radiator is formulated. Using appropriate similarity transformation and shooting quadrature coupled with Runge–Kutta–Fehlberg integration scheme, the model boundary value problem is tackled numerically. A parametric study of the entire flow regime is carried out to illustrate the effects of the pertinent parameters on the velocity, temperature, skin friction coefficient and the local Nusselt number. It is clear that CuO–EG nanofluids lead to a rapid decrease of temperature at the boundary layer.

Mixtures of ethylene glycol and water are used in cooling the engines in automotive applications. To avoid the two-phase flow in the engine, the mixture is subcooled in the radiator before entering the engine block. Heat transfer is therefore essentially under subcooled flow boiling conditions. Very little information is available in the literature on the subcooled flow boiling characteristics of this mixture, and there is no predictive method established in this region. Satish G. Kandlikar et.al, focuses on obtaining experimental heat transfer data for mixtures of ethylene-glycol (0 to 40 percent mass fraction, limited by the maximum allowable temperature in the present setup) and water in subcooled flow boiling region. The experimental setup is designed

to obtain local heat transfer coefficient values over a small circular aluminum heater surface, 9.5-mm in diameter, placed at the bottom wall of a rectangular channel 3- mm x 40-mm in cross-section. The applicability of the available model for subcooled flow boiling of pure liquids to the mixtures is examined.

2.1 OBJECTIVES

- Development of an experimental set up to demonstrate battery cooling using ethylene glycol through corrugated channel.
- Conducting experiments by varying mixture ratio of ethylene glycol and water.
- Develop a numerical model using Ansys Fluent to study the heat transfer and fluid flow during cooling
- Validate the numerical model with experimental data.
- Find out an optimum channel geometry and mixture ratio which can deliver maximum cooling.

2.3 METHODOLOGY

1. Experiments using ethylene glycol in corrugated channel.
2. Experiments using ethylene glycol and water by varying mixture ratio.
3. Numerical Simulation
 - Using water and smooth channel
 - Using water and corrugated channel
 - Using ethylene glycol and corrugated channel
 - Using different combinations of ethylene glycol and water in corrugated channel.
4. Comparison between experimental and numerical values.
5. Finding out an optimum mixing ratio which can provide better cooling.

CHAPTER 3

BATTERY THERMAL MANAGEMENT SYSTEMS

3.1 BATTERY THERMAL MANAGEMENT SYSTEMS

Basic types of BTMS are,

1. Air cooling
2. Liquid cooling
3. Direct Refrigerant cooling
4. Phase change material cooling
5. Thermoelectric cooling
6. Heat pipe cooling

3.2 AIR COOLING

Due to their inexpensive cost, straightforward design, light weight, ease of maintenance, and lack of leakage problems in comparison to other cooling systems, air cooling systems are among the most popular BTMSs in EVs. By constantly moving air between the batteries in this arrangement, heat is removed. According to estimates, air cooling might manage typical electric vehicle driving circumstances if heat rejection from the battery pack is less than about 4 kW.

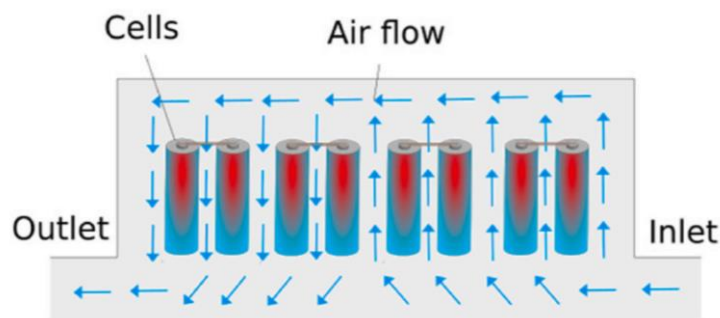


Fig 3.1: Air cooling method (G. Offer et al, 2020).

The two primary categories of air-cooling systems are active air cooling (driven convection) and passive air cooling (natural convection). The forced air flow into the battery pack improves convective heat transfer, hence the active air-cooling system has

a higher cooling efficiency. However, this requires more parasitic energy from fans, as well as more weight and space associated with fan ducts and manifolds.

On the other hand, passive air-cooling systems that just use natural air flow have no parasitic power usage and hence perform less well at cooling. As a result, active air-conditioning systems represent a higher share of the market and have been used in a number of applications, including the cooling systems for the Nissan e-NV, Toyota Prius, and Honda Insight. Recent years have seen improvements in the efficiency of air-cooling systems. Three key factors—cell arrangement, air flow channel, and flow rate—are at the centre of this.

3.3 LIQUID COOLING

It is a cooling system in which water is used as the coolant for the purpose of cooling the battery. Liquid cooling is the most commonly used cooling system due to its convenient design and good cooling performance. Compared with air, liquids have higher specific heat capacity as well as better thermal conductivity. Given the limitations of air-cooling systems, liquid cooling is an alternative route for large scale EV BTMSs. Dielectric liquid cooling or direct-contact liquid which can contact the battery cells directly, such as mineral oil. The other is conducting liquid or indirect-contact liquid which can only contact the battery cells indirectly, such as a mixture of ethylene glycol and water. Depending on the different liquids, different layouts are designed. For direct-contact liquid, the normal layout is to submerge modules in mineral oil. For indirect-contact liquid, a possible layout can be either a jacket around the battery module, discrete tubing around each module, placing the battery modules on a cooling/heating plate or combining the battery module with cooling/heating fins and plates. Between these two groups, indirect contact systems are preferred to achieve better isolation between the battery module and surroundings and thus better safety performance. The studies on the liquid cooling system have always been fixated at the development of the physical design of the cooling plate and its channels and by targeting the parameters like; coolant pressure drop across the channels of the cooling plates and cell core temperature different designs are fabricated. According to the previous research on the geometric development of the cooling plate, the highest cooling performance was achieved by channelled cooling and it also showed the minimum power consumption as

compared to other methods. But channelled cooling is not ideal for temperature consistency due to the comparatively long path of flow.

The path of the heat transfer from the bottom of the battery to the cooling plate highly contribute to the thermal resistance of the battery pack structure, several modified pack designs are devised to enhance the cooling performance such as Thickened cooling fin design, Sandwich cooling plate design and the Interspersed cooling plate design. To optimize the structural design of the practical and large-scale battery thermal management system for electric vehicles. A thermal model for the indirect fin-cooling battery pack is developed, type D-2 was proposed as an alternative design for BTMS. It improved the ratio of equivalent heat conductance to the system volume by 64% and the total pressure drop is increased by 19% and the maximum temperature difference was reduced by 5.4°C.

3.3.1 Indirect Cooling System

Indirect cooling is one of the most widely used EV BTMS due to its ability to maintain a good uniform pack temperature distribution, favourable specific heat capacity, and good thermal control. Mixtures of water and ethylene glycol are common coolants. Normally, the coolant flows along channels of pipes or cooling plates, carrying the rejected heat out of the battery pack. Based on the cooling channels' position relative to the batteries, indirect cooling systems can be divided into bottom cooling and side cooling. The side plate system enabled a lower maximum temperature and better temperature uniformity than the bottom cooling plate system due to the more comprehensive contact with the battery.

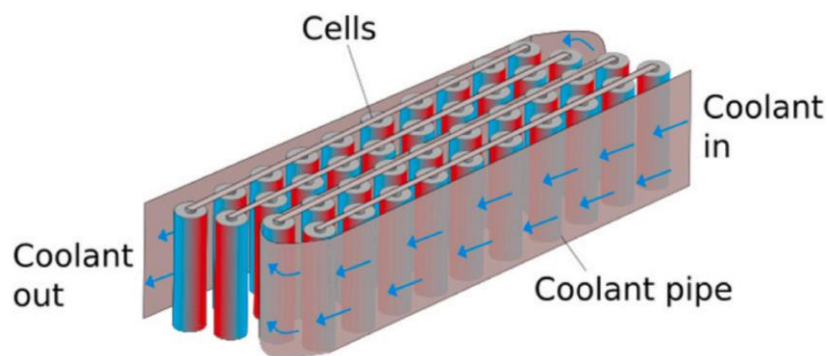


Fig 3.2: Indirect Liquid Cooling (G. Offer et al, 2020)

Heat pipes are another form of indirect liquid cooling. These offer extremely high thermal conductivities through the evaporation and condensation of a 2-phase working fluid contained within a thermally conductive sealed pipe. Working fluids such as water and acetone are common, however, whilst superior heat transfer can be achieved over pure metallic heat sinks, making effective thermal contact was identified as a key challenge. Furthermore, many studies suggest that forced air cooling would be needed to condense the working fluid and ensure suitable temperature operation due to the limited heat capacity of the heat pipes. Various hybrid systems have been proposed which combine the high specific heat of PCMs and the high thermal conductivity of heat pipes.

Compared with air cooling systems, the complexity of the whole system is much higher due to the additional components (e.g., chiller and radiator) and complicated piping. This results in more potential sources of failure, additional weight and coolant leakage issues. Beyond this, the thermal resistance between the cell and coolant is often increased due to the existence of cooling tubes/plates as well as imperfect thermal contact, which might be an application barrier to ultrafast charging conditions (300 kW).

3.3.2 Tab cooling

Recent progress on pouch cell tab cooling has been proven as a promising way to achieve high temperature uniformity. For instance, Hunt et al. [18] compared the electrochemical performance between surface cooling and tab cooling with a 5 Ah pouch cell with tabs on opposite ends of the cell. The two cooling methods exhibited similar capacity loss under lower current operation of C/20 but at the higher rate of 6C this accessible capacity loss was more apparent. The loss of useable capacity for the pouch cell with surface cooling was 9.2%, while the pouch cell with tab cooling demonstrated a stable performance with only 1.2% loss of useable capacity. After 1000 cycles, surface cooling caused a capacity loss which was three times higher than cell tab cooling. The enhanced electrochemical performance of tab cooling was attributed to more homogeneous temperature distribution, leading to more homogenous current distributions between cell layers.

Considering most use cases in real-world application are still dominated by the surface cooling method, tab cooling may be promising due to its ability to impose favourable temperature gradients, and potential to extend cell lifespan. Nevertheless,

two bottlenecks still needed to be taken into account before tab cooling can be upscaled. Firstly, tab cooling may not be as effective in controlling the maximum cell temperature, especially for harsh utilization conditions. Secondly, the practicalities of tab cooling systems designs are still being debated, with the ease of system manufacturability a key issue. The tab should be electrically isolating but most cooling tubes are aluminium. How to accelerate the heat transfer rate while maintaining the tabs in an insulation environment is still, therefore, under debate. Immersion may provide a more simplistic version of tab cooling as the tabs can be inadvertently cooled while the cell is immersed without the need of a complex tab cooling system.

3.3.3 Immersion cooling

As one of the emerging cooling technologies in recent years, direct liquid cooling, which is also called immersion cooling, has attracted considerable attention for electronic devices and in the EV industry. In this system, the battery is submerged into a non-conductive dielectric fluid, thus making direct contact with the cell. Candidate dielectric fluids have included: hydrocarbon oils, silicone oils and fluorinated hydrocarbons. This unique way of cooling brings several advantages. Firstly, immersion cooling has the potential to provide the best pack and cell temperature uniformity among all the cooling methods. This is because all battery surfaces are in the fluid, providing a homogeneous, high heat capacity thermal transport path for heat rejection. This direct contact with the cell surfaces further reduces the thermal contact resistances experienced in indirect cooling systems. Immersion cooling simplifies the system design and reduces the system complexity. Furthermore, the suppression of thermal runaway is usually observed for immersion cooling systems as some of the dielectric fluids are also flame-retardants, enhancing the safety of the LIB pack.

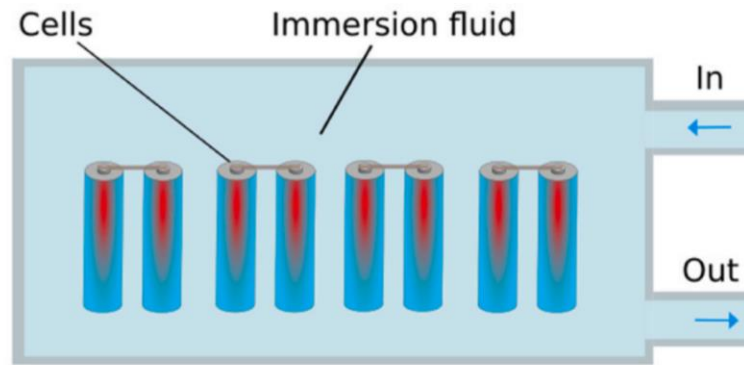


Fig 3.3: Immersion Cooling (G. Offer et al, 2020)

However, disadvantages include: the added complexity/cost of condensing evaporated vapour, potentially higher pumping losses in high viscosity fluids, high cost of the fluid, material compatibility issues and additional weight of the fluid

3.3.4 Direct refrigerant cooling

Similar to active liquid systems, a direct refrigerant system (DRS) consists of an Air Conditioning loop, but Direct Refrigerant System uses refrigerant directly as heat transfer fluid circulating throughout the battery pack.

3.3.5 Phase change material

Phase change materials (PCMs) have also received significant attention in BTMSs. The latent heat associated with the material, as it undergoes a phase transition during heating, affords it with favourable specific heat capacity characteristics. This can come in different forms with the main classifications including: solid-liquid and liquid-gas PCMs. When selecting a PCM for a BTMS, the melting point, specific heat capacity and thermal conductivity are often key figures of merit. One of the main organic PCMs used is paraffin, with the length and formation of carbon within the chains giving the paraffin different physical and chemical properties. The melting point of paraffin is dependent on the length of the straight chain alkanes and the mixture combination of alkane chains. The specific heat capacity of paraffin wax ranges from 2140 to 2900 J/kg.K which is higher than many immersion coolants.

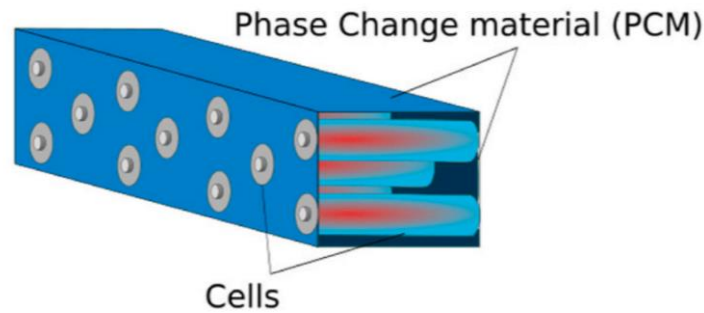


Fig 3.4: Phase Change Material (G. Offer et al, 2020).

Phase Changing Materials are classified into three major categories: Organic (paraffin compounds, non-paraffin compounds), Inorganic (Salt hydrates metallics), and Eutectic mixtures. The organic and salt hydrates phase-changing materials are favourable with applications with temperatures less than 10°C (e.g., Li-Ion Batteries). Eutectic mixtures can be used for temperatures up to 25°C. Organic materials have latent heat of fusion in the spectrum of 128 to 200KJ/Kg while Inorganic compounds have a range of 250-400KJ/Kg. Organics PCM is generally branched into two sub-categories: paraffin and nonparaffin. Paraffin is examined to be safe, chemically stable, dependable, and inexpensive. Furthermore, they have low volumetric expansion through the phase transition and have low transition pressure. Paraffin is composed of chains of alkanes whose chemical structure and formula are $\text{CH}_3(\text{CH}_2)_m\text{CH}_3$ and $\text{C}_n\text{H}_{2n+2}$. Generally, paraffin phase-changing materials have their melting temperature and latent heat grow logarithmically with an increase in the number of carbon atom. The main disadvantage of paraffin phase-changing material is a low thermal conductivity which is in the range of 0.15W/mK to 0.12W/mK. Non-Paraffins can subcategorized as esters, alcohols, glycols, and fatty acids. Normally non-paraffins organic PCMs are distinguished by the high heat of fusion, nonflammability, low thermal conductivity, wild toxicity, and instability at high temperature. Moreover, fatty acids are the most important sub-category of inorganic PCMs. They have high heat of fusion compared to paraffin and no issue of thermal hysteresis and subcooling during freezing processes. The chemical structure and formula are $\text{CH}_3(\text{CH}_2)_m\text{COOH}$ and $\text{C}_n\text{H}_{2n}\text{O}_2$.

The latent heat and energy density of fatty acids increases with an increase in melting temperature and the normal range are 150KJ/Kg to 200KJ/Kg and 35kWh/m³

to 51kWh/m³ respectively. The thermal conductivity of fatty acids is very low, i.e., from 0.14K/mK to 0.17K/mK. Thermal diffusivities of fatty acids are in the range of 7.5m²/s to 10-2m²/s. Advantages of non-paraffin PCMs are good chemical stability, non-toxic, low volumetric expansion, compatibility with storing materials, high latent heat and energy density, no effect of sub-cooling and phase segregation. Disadvantages of non-paraffin PCMs they are more expensive when compared with paraffin and salt hydrates. The cost of non-paraffin PCM is approximately 2 to 2.5 times of paraffin and more than that when compared with salt hydrates. Inorganic PCMs are sub-categorized into two parts: salt hydrates and metallics. Salt hydrates are a mixture of alloys of inorganic salts (AB) and water (nH₂O), to form a compound with a corresponding chemical formula as AB.(nH₂O). In this type of PCM, melting/solidification is a dehydration/hydration of the salt. This causes an issue of salt hydrates i.e., non-congruent or sedimentation processes during melting. This is caused due to dehydrated salts being heavier than water and tend to sediment at the bottom of the container. When hydration needs to be activated, the system is distinguished by areas of dissimilar salt concentrations, and therefore complete hydration is not possible. However, resolutions for this problem have been already found, for instance, mechanical stirring, encapsulating the PCM to avoid the separation of dehydrated salt from its water content, and also adding special thickening materials. The other issue with salt hydrates is their super-cooling, because of the low nucleation property of the material. It means that the nucleation rate of salt hydrates is very low at transition temperature and the material needs to be super-cooled before nucleation is naturally activated in the salt hydrates. It means that it releases the thermal energy which is stored in the material at a much lower and thus decreasing the exergy efficiency of the heat storage system. There is little evidence that the addition of a nucleation agent or even injecting nuclei can activate the cooling process. Altogether, the advantages of salt hydrates are high latent, high thermal conductivity, low volumetric expansion during melting, low level of toxicity and corrosivity (compatible with plastics), and cheap when used in pure form. The last classification of inorganic PCMs is metallics. Metallics are low temperature melting metals. They have high volumetric energy density but because of high density; they have low specific energy density. PCMs have high thermal conductivity, so it does not need thermal conductivity enhancement. The other category is eutectic mixtures. Eutectic mixtures of 2 or more PCMs, which at the specific configuration, melt at singular temperature.

Phase Changing Material is a matter that absorbs and releases thermal energy in course of melting and freezing. When a PCM freezes, it releases abundant energy in the form of latent heat at a relatively constant temperature. On the contrary, when such material melts it absorbs an enormous amount of heat from the environment. PCMs recharge as the surrounding temperature fluctuates, making them ideal for a variety of everyday application that requires temperature control. PCMs have been developed for covering a wide span of temperatures from -4°C to more than 15°C . They normally store 5 to 14 times more heat per unit volume than materials such as Water, Mansory, or Rock. Amid many heat storage alternatives, PCMs are attractive because they offer high-density energy storage and store heat within a narrow temperature range.

3.3.6 Thermoelectric cooling

Thermoelectric coolers which are used in battery thermal management systems are a comparatively new technology in the field of electric vehicles. Their advantages are strong cooling capacities and reliable working potential and have increasingly gained attention for integration into battery thermal management system. The main issue with the air- and water-cooling method is the cooling effect can be very limited under certain circumstances. A thermoelectric module is a solid-state energy converter that consists of a bunch of thermocouples connected in series and thermally in parallel.

A thermoelectric cooler (TEC) is based on the conversion of voltage to temperature difference. It refers to all of the transformation processes from heat to electricity and vice-versa. It operates according to the Peltier effect. The effect creates a temperature variance by carrying heat between two electrical junctions. A voltage is applied across joined conductor to create an electric current. When the current flows through the two conductors, heat is removed at one of the junction and cooling occurs. Heat is deposited at the opposite junction. The foremost relevance of the Peltier effect is to cool. The Peltier effect can also be used for heating or control of temperature.

Thermoelectric cooling has several advantages over other cooling methods like static device, no internal chemical reaction, noise-free, longer operation, no emission of hazardous gases, and minimum maintenance cost. Disadvantages of TEC are low efficiency and additional power requirement which limits their commercial application.

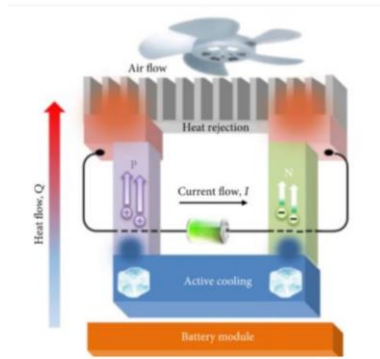


Fig 3.5: Thermoelectric cooling (Chaitanya Pandya et al, 2021)

TECs application revolves around two main aspects i.e., converting heat to electricity and electricity to heat. There are numerous applications of TEC. The main application of TECs is its use in cooling Li-Ion batteries, microprocessor of high configured computer and for building air conditioning system. TECs have been also used recently in portable refrigerators, portable air conditioners and automobile cooling. The most promising application of TECs is integrating it with PCMs for BTMSs to make a passive system into semi-passive system and thus increasing the efficiency of the BTMS.

3.3.7 Heat pipe cooling

It's a passive cooling system which is basically a sealed tube filled with refrigerant and it absorbs heat by vaporizing the refrigerant from the hot side and it removes heat into surrounding by condensing the refrigerant back to liquid, form on the cold side and then flows back. A partial vacuum is maintained in the casing of the heat pipe and to increase the heat transfer rate of heat pipes a capillary structure is used within the heat pipes which increases the surface temperature. The heat pipe may use water or any refrigerant as the coolant and this cycle repeats again and again. The battery acts as a heat source and sits below the heat pipe (on the evaporating side) and the cooling fins acts as the heat sinks on the heat pipe (on the condensing side). According to experiments, a reduction of 30% in the thermal resistance is found for heat pipe cooling system under natural convection as compared to without heat pipe. A thermal resistance reduction of 20% under low air velocity convection is possible. The main problem with this cooling is the safety of the system which can be a concern in the case of an

emergency; a short circuit can happen due to the coolant leakage on the battery cells which can cause a failure of the vehicle and can be fatal. Also, the capillary tubes require a minimum diameter to maintain an adequate pressure drop and avoid blockage.

CHAPTER 4

EXPERIMENTAL METHOD

4.1 EXPERIMENTAL SETUP

The experimental apparatus used in this study consists essentially of a horizontal channel in which Ethylene Glycol and water mixture flows over a localized heater. The experimental setup consists of a corrugated channel, heat exchanger, a pump and a temperature display unit. The test section is fabricated from four copper plates of height 30mm and length 600mm. The plates are bend into corrugations so that it can accommodate a cell in between the corrugations. The angle of bend of the channel is 53.5° with the tangent. After bending, the plates are joined to form the channel.



Fig 4.1 Corrugated channel

The heat exchanger used for the experiment is an externally insulated box. Coil windings are incorporated inside the heat exchanger. The Ethylene glycol and water mixture is passed through the coil. The box is filled with cold water at 18°C . Aluminium plates of 6mm thickness are joined to form a rectangular box of dimensions $15\text{cm} \times 15\text{cm} \times 15\text{cm}$. Two holes are provided in the box for incorporating the coil inside it. Coil windings are made using copper tube of diameter 8mm. The copper tube is bent into a helical shape with pitch 4cm and height 12.5cm. The coil is welded inside the aluminium box. The coil and the aluminium box are checked thoroughly for any leaks. The outer part of the heat exchanger box isothermally insulated from all sides. This is

done to prevent the heat from entering the chamber and increasing the temperature of the cold water inside the chamber. A centrifugal pump as per the need is arranged to create flow. Then the corrugated channel, heat exchanger and pump are connected together using silicon tubes. The thermocouple is then fixed at the battery surface to measure the temperature of the surface at different instances.

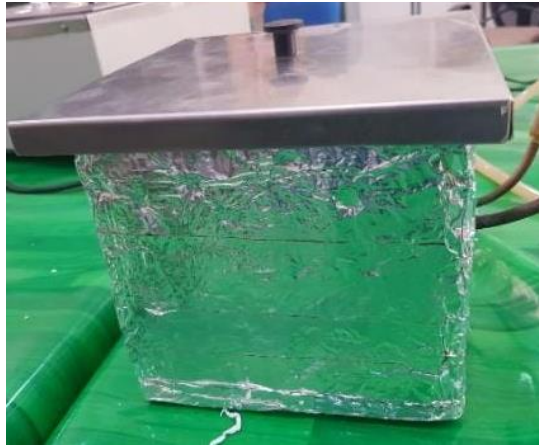


Fig 4.2 Heat exchanger used for experiment

For simulating the working conditions of a lithium-ion battery, Nichrome wire and thermal grease paste is used. The thermal grease paste is moulded into a cylindrical shape with a hole in the centre for incorporating the nichrome wire. The cylinder block is of the size 18mm diameter and 65mm height. Five such blocks are made and is placed in the channels such that the surface of the cylinder is in perfect contact with the corrugations in the channel. The nichrome wire is bend in a manner such that it passes through the centre of all the five-cylinder blocks. The two ends of the nichrome wire are free and is provided with provisions to connect to the electric supply. The voltage can be adjusted using an auto transformer.



Fig 4.3 Pump used for experiment.



Fig 4.4 Experimental setup.

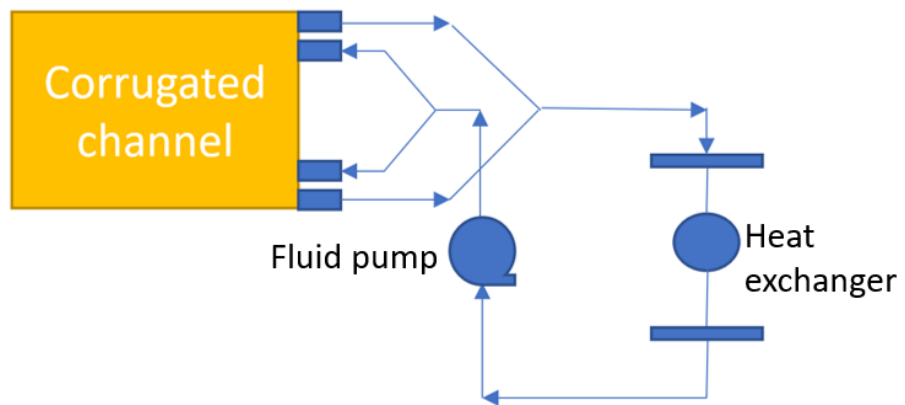


Fig 4.5 Schematic diagram of experimental setup.

Temperature is measured using thermocouple of T-type and it is connected to a temperature display unit. Using this type of thermocouple, can measure temperature from -200°C to 200°C . It consists of positive leg made of a Copper wire and negative leg made of Constantan (Cu & Cu-Ni) alloy wire.

4.2 EXPERIMENTAL PROCEDURE

The experiments were performed with pure water and mixture of ethylene glycol and water with different ratio. For the initial test runs, distilled water was used as the working fluid in the system. After each run, ethylene glycol was added systematically to yield the desired concentration for the next run. At the beginning of the data collection sessions, the water bath was filled and the pump and the heater were turned on. The test section was maintained at atmospheric pressure.

The power to the heater and the flow rate were set to the desired values. The system reached steady state conditions within 20 minutes. Temperature along the heater length and inside the flow channel were recorded after 30 minutes. From the temperature measurements in the heater at known locations, the temperature of the heated surface is determined. The tests were performed by powering the heaters and systematically increasing the heat flux between the successive runs.

4.3 EXPERIMENTS USING WATER IN CORRUGATED CHANNEL

The first run used 500mL of water. The water was measured using measuring cylinder. The water was filled into the experimental setup using the opening provided. While filling the fluid in the setup, it should be ensured that there were no air gaps in the tubes as it may cause inefficiency.

For conducting the experiments, first proper electric supply was given to the nichrome wire. This was done using the auto transformer. When electric current was passed through nichrome wire, it began to generate heat. This heat generated was conducted by the thermal paste which is moulded as the cylinder block. The conducted heat reaches the outer surface of the corrugated channel. The temperature of the channel increases and was measured and monitored using a thermocouple. The experiment was started only after the temperature of channel reached 40°C. To begin the experiment, the centrifugal pump is switched on. When the pump is switched on, the fluid starts flowing in the circuit. The pump was adjusted so that it provided a velocity of 0.118m/s to the fluid. The fluid was allowed to flow for a few minutes to reach steady state before the readings were taken. When the fluid reaches steady state, the readings were taken.

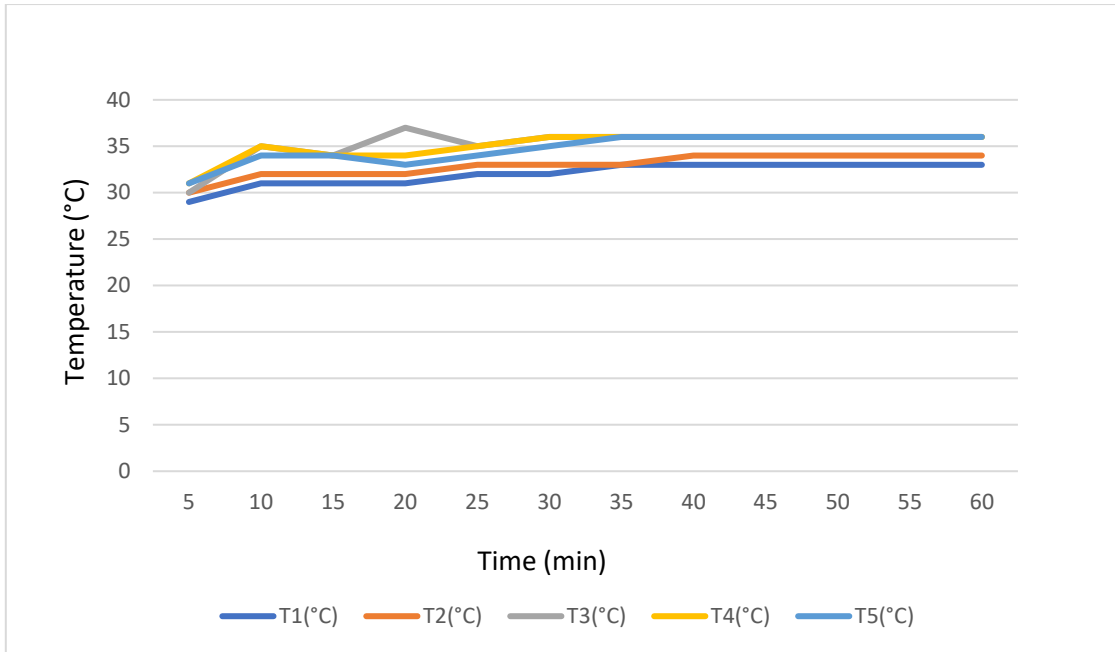


Fig 4.6 Experiments using water in corrugated channel

4.4 EXPERIMENTS USING ETHYLENE GLYCOL IN CORRUGATED CHANNEL

The cooling medium was 500mL of ethylene glycol. The procedure was similar as the above experiment.

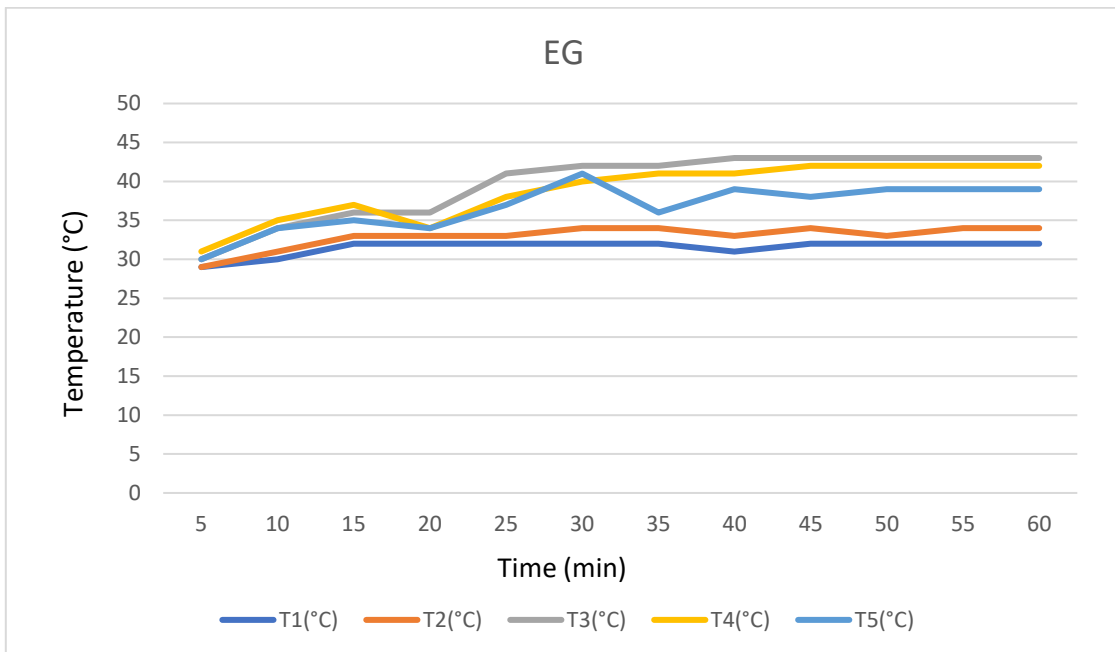


Fig 4.7 Experiments using EG in corrugated channel

4.5 EXPERIMENTS USING ETHYLENE GLYCOL AND WATER IN CORRUGATER CHANNEL IN THE RATIO 40:60

Ethylene glycol and water are used in the ratio 40:60. 500mL of the fluid was prepared. So, the volume of ethylene glycol and water required were 200mL and 300mL respectively. The fluids were added to a beaker separately after measuring them. All other procedure for the experiment is same as the above.

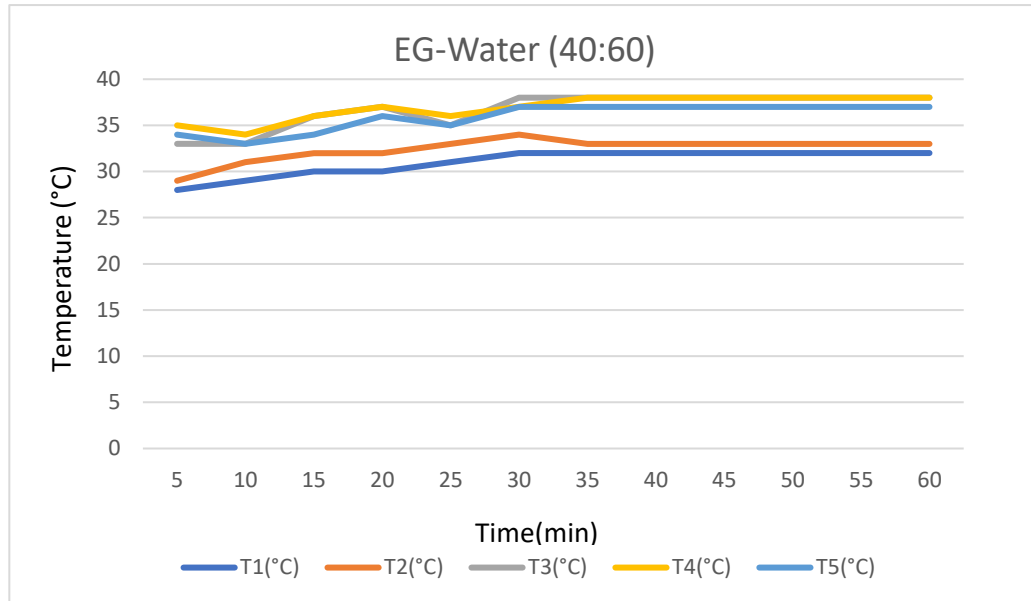


Fig 4.8 Experiments using EG-water (40:60) in corrugated channel

4.6 EXPERIMENTS USING ETHYLENE GLYCOL AND WATER IN CORRUGATER CHANNEL IN THE RATIO 50:50

Ethylene glycol and water are used in the ratio 50:50. 500mL of the fluid was prepared. So, the volume of ethylene glycol and water required were 250mL and 250mL respectively. The fluids were added to a beaker separately after measuring them. All other procedure for the experiment is same as the above.

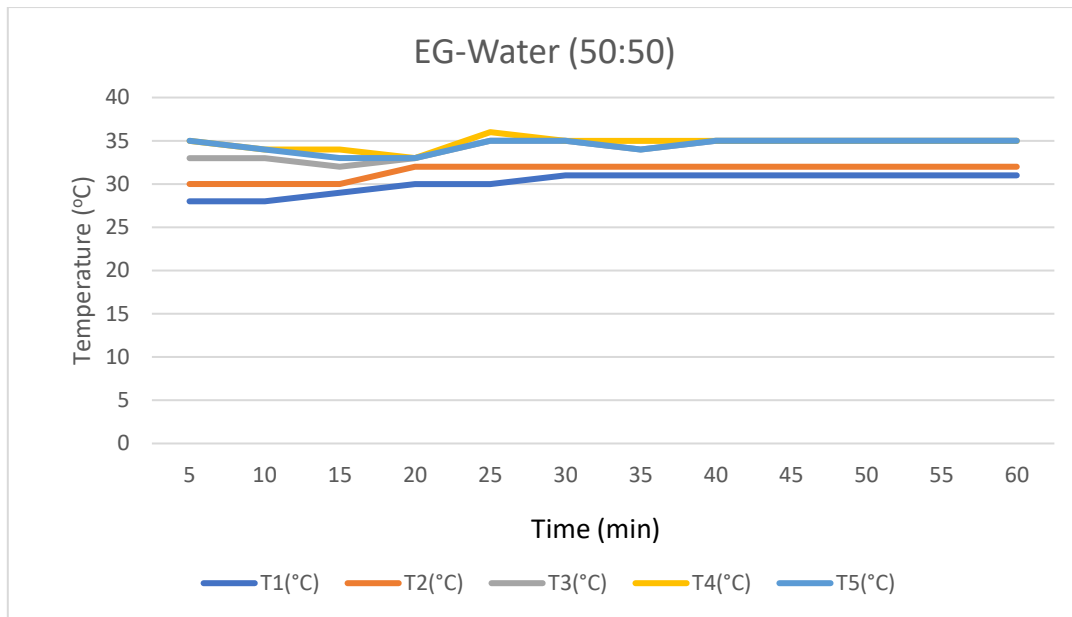


Fig 4.9 Experiments using EG-water (50:50) in corrugated channel

4.7 EXPERIMENTS USING ETHYLENE GLYCOL AND WATER IN CORRUGATED CHANNEL IN THE RATIO 60:40

Ethylene glycol and water are used in the ratio 60:40. 500mL of the fluid was prepared. So, the volume of ethylene glycol and water required were 300mL and 200mL respectively. The fluids were added to a beaker separately after measuring them. All other procedure for the experiment is same as the above.

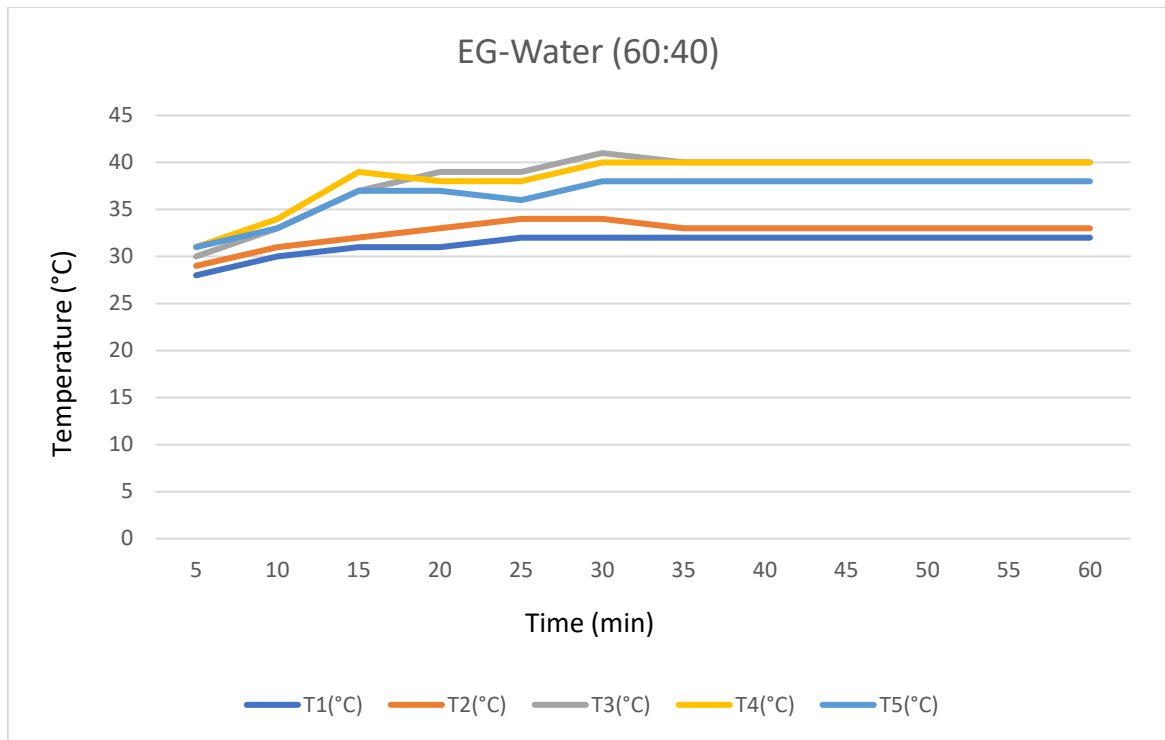


Fig 4.10 Experiments using EG-water (60:40) in corrugated channel

CHAPTER 5

CFD SIMULATION

Different approaches to study flow problems are conducting experiments in physical setup like wind tunnels, theoretical solutions of governing equation and numerical solution of governing flow equations (CFD).

Experiment is the most realistic way of solving a flow problem. Wind tunnels, measuring instruments, model fabrication and instrumentation used for experiments are costly. Many quantities in experiments may be difficult to measure. Simultaneous matching of similarity parameters may be difficult, if not possible and also scale effects. Theoretical solution is the exact or analytical solution of governing equations. It is often restricted to simple geometries. Assumptions which may not very well cater to real world situations. When theoretical solutions do not exist, use CFD. Experiments may be difficult to conduct or facilities do not exist. In CFD we can get resolution numerical data in space and time. Flow parameters can be changed easily without additional expense, which could be very difficult in experiments.

CFD (Computational Fluid Dynamics) is the analysis of systems involving fluid flow, heat transfer and other associated transport phenomena like chemical reactions by means of computer-based simulation.

5.1 APPLICATIONS OF CFD

5.1.1 Aerospace / Mechanical Engineering

- Aerodynamics: - Aircrafts and its components – fixed wing, rotary wing/ UAV-MAV / Spacecraft / ground vehicle – obtain velocity, pressure, flow structures, forces and moments.
- Power plants and Propulsion: - Combustion in IC engines and gas turbines.
- Turbomachinery: - Flows inside rotor stator blade passage in fans, propellers, compressors, gas turbines, wind turbines.
- HVAC systems in automobiles, aircrafts.
- Heat exchangers.

5.1.2 Ocean and Marine Engineering

- Hydrodynamics of ships and submarines.
- Hydrodynamics of off-shore structures: floating structures, underwater pipe lines.
- Hydrology and Oceanography: - flows in rivers, estuaries, oceans.

5.1.3 Environmental Engineering

- Distribution of pollutants and effluents in the atmosphere.
- Meteorology: weather prediction.
- External and internal environment of buildings: wind loading and heating /ventilation.

5.1.4 Electrical and Electronic Engineering

- Cooling of equipment including microcircuits, chips.

5.1.5 Chemical process Engineering

- Mixing and separation, polymer moulding

5.1.6 Biomedical Engineering

- Blood flows through arteries and veins.

5.2 STEPS INVOLVED IN PERFORMING CFD SIMULATION OF A FLOW PROBLEM

- Problem definition and objectives of performing CFD simulation.
- Choice of governing equations defining the level of the approximation to reality that we would like to simulate.
- Domain definition and discretization - Grid generation. **(Pre-processing step)**.
- Methods of solution of governing equation often partial differential equations based on suitable initial and boundary condition: e.g., Finite Difference Method (FDM), Finite Volume Method (FVM), Finite Element Method (FEM). Most often, analytical solution does not exist for the problem. Stability and accuracy of the numerical scheme must be analysed before using it for flow simulation. **(Flow solver setup)**.
- Analysis of results and validation **(Post processing setup)**.

5.2.1 Finite Difference Method (FDM)

The simplest numerical technique to apply for the solution of the heat/diffusion equation is the finite difference method. The basic idea behind the method is to replace the various derivatives appearing in the mathematical formulation of the problem with suitable approximations on finite difference mesh of nodes. The simplest derivation of finite difference formulae makes use of the Taylor series. The final set of linear algebraic equations is solved by any numerical techniques.

5.2.2 Finite Element Method (FEM)

The finite element method subdivides the calculation domain into elements, such as triangular rectangles, tetrahedral or rectangular parallelepipeds elements. These elements are considered interconnected at specified joints called nodes. Here the variation of field variable inside a finite element can be approximated by a simple function. These approximating functions are defined in terms of the values of the field variables at the nodes. When field equations are written for the whole continuum the new unknowns are at the nodal points. By solving the field equations, which are generally in the form of matrix equations, the node values of the field's variables will be known. Once these are known, approximating functions define the field variable through the assemblage of elements.

5.2.3 Finite Volume Method (FVM)

This is the classical or standard approach used most often in commercial software and research codes. An alternative discretization method is based on the idea of regarding the computation domain as subdivided into a collection of finite volumes. In this view, each finite volume is represented by a line in 1D, an area in 2D, and a volume in 3D. Nodes, located inside each finite volume, become the locus of computational values. In rectangular Cartesian coordinates in 2D the simplest finite volumes are rectangles. For each node, the rectangle faces are formed by drawing perpendiculars through the midpoints between contiguous nodes. Discretization equations are obtained by integrating the original partial differential equation throughout each finite volume. This method is easily extended to nonlinear problems. The solutions of the algebraic equations are obtained by iterative methods.

One advantage of this method over FDM is that it does not require a structured mesh - although a structured mesh can be used. The FVM can solve problems on irregular 18 geometries. Furthermore, one advantage of this method over FEM is that it can conserve the variables on a coarse mesh easily. This is an important characteristic of the fluid problem.

5.3 WORKING OF A CFD CODE

CFD codes are structured around numerical algorithms that can tackle fluid flow problems. To provide easy access to the solving power, all commercial CFD packages (Phoenics, Flow3D, and Star CD) include sophisticated user interfaces to input problem parameters and to examine the results. All codes contain 3 main elements.

- Pre-Processor
- Solver
- Post-Processor

5.3.1 Pre-processor

Pre-processing consists of the input of a flow problem to a CFD program using an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve

- Definition of the geometry of the region of interest: the computational domain
- Grid generation: the sub-division of the domain into several smaller, non-overlapping sub-domains known as a grid of cells or mesh (control volumes or elements).
- Selection of the physical and chemical phenomena that need to be modelled.
- Definition of fluid properties.
- Specification of appropriate boundary conditions at cells that coincide with or touch the domain boundary.

The solution to a flow problem is defined at the nodes inside each cell. The accuracy of a CFD solution is governed by the number of cells in each grid.

5.3.2 Solver

There are three distinct streams of numerical solution techniques: finite difference, finite element, and spectral methods. In outline, the numerical methods that form the basis of the solver perform the following steps.

- Approximation of the unknown flow variables using simple functions.
- Discretization by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.
- Solution of the algebraic equations.

The main differences between the three separate streams are associated with how the flow variables are approximated and with the discretization processes.

5.3.3 Post-processor

As in pre-processing a huge amount of development work has recently taken place in the post-processing field. With the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools. These include,

- Domain geometry and grid display
- Vector plots
- Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking
- View manipulation (translation, rotation, scaling, etc.)
- Colour postscript output.

5.4 COMPUTATIONAL FLUID DYNAMICS SIMULATION

The design, scale-up, and running of unit operations in chemical process industries rely heavily upon empiricism and correlations of overall parameters for non-ideal or non-equilibrium conditions. Many equipment designs in use are based on the experience of experts applying rules of thumb, resembling art more than science. Processes that are

sensitive to local phenomena and reactant concentrations are often difficult to design or scale up because the design correlations do not take local effects into account. Non-idealities introduced by scaling up of lab or pilot scale equipment are difficult, if not impossible to predict accurately.

Researchers, equipment designers, and process engineers are increasingly using computational fluid dynamics (CFD) to analyse the flow and performance of process equipment, such as chemical reactors, stirred tanks, fluidized beds, cyclones, combustion systems, and spray dryers, pipeline arrays, heat exchangers and other equipment. CFD allows for in-depth analysis of the fluid mechanics, local effects, and chemistry in these types of equipment such as turbulence and combustion. CFD can be used when design correlations or experimental data are not available. It provides comprehensive data that are not easily obtainable from experimental tests. It highlights the root cause, not just the effect, and many ‘what if’ scenarios can often be analysed in a short time. This method reduces the scale-up problem because the models are based on fundamental physics and are scale-independent.

CFD is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm. It is the merger of the classical branches of theoretical and experimental science, with the infusion of the modern element of numerical computation. The results of CFD analyses are relevant engineering data used in conceptual studies of new designs, detailed product development, troubleshooting, and redesign. In many cases, CFD results in better insight, improved performance, better reliability, more confident scale-up, improved product consistency, and higher plant productivity.

The progress of CFD during the last fifty years has been extraordinary. Much of this progress has been driven by the phenomenal increases in digital computing speed. The continual and exponential increase in computing power improved physical models in many CFD codes, and better user interfaces now enable non-experts to use CFD as a design tool on a day-to-day basis. As a consequence, CFD has progressed from the domain of mainframe to the high-end engineering workstation and even to laptop PCs. This power of digital computing has transformed research and engineering, especially in fluid mechanics, just as it has in virtually all fields of human endeavours.

5.5 PHASES OF MODELLING AND SIMULATION

There has been a long history of efforts to establish the basic concepts and terminology in modelling and computer simulation. The identification of the fundamental issues and debates began two decades ago in the operation research community, long before there was such concern in the CFD community. The terms model, modelling, and simulation are used in a wide range of disciplines. Consequently, these terms have a range of meanings that are both context-specific and discipline-specific. A model is a representation of a physical system or process intended to enhance our ability to understand, predict, or control its behaviour. Modelling is the process of construction or modification of a model. Simulation is the exercise or use of a model. The basic phases of modelling and simulation have been identified by the operation research community. It identifies two types of models: a conceptual model and a computer model. The conceptual model is composed of all the information, mathematical modelling data, and mathematical equations that describe the physical system or process of interest. The conceptual model is produced by analysis and observations of the physical system. In CFD, the conceptual model is dominated by partial differential equations. The computer model is an operational computer program that implements a conceptual model. Modern terminology refers to the computer model as computer code.

Although CFD simulations are widely conducted in industry, government, and academia, there is presently little agreement on procedures for assessing their capability. There is no fixed level of credibility or accuracy that applies to all CFD simulations. The accuracy level required of simulations depends on the purposes for which the simulations are intended to be used.

The two main principles that are necessary for assessing credibility are verification and validation. Verification is the process of determining if a computational simulation accurately represents the conceptual description of the model and the solution to the model, but no claim is made of the relationship of the simulation to the real world. Validation is the process of determining if a computational simulation accurately represents the real world from the perspective of the intended uses of the model. The definition of verification and validation also stresses the evaluation of accuracy. In verification activities, accuracy is generally measured with the respect to

benchmark solutions of simplified model problems. In validation activities, accuracy is measured concerning experimental data, which represent reality.

Uncertainty and error can be considered as the broad categories that are normally associated with the loss in accuracy in modelling and simulation. Uncertainty is defined as a potential deficiency in any phase or activity of the modelling process that is due to a lack of knowledge. Lack of knowledge is commonly caused by incomplete knowledge of a physical characteristic or parameter. Lack of knowledge can also be caused by the complexity of a physical process, for example in the case of turbulent combustion. Error is defined as a recognizable deficiency in any phase or activity of modelling and simulation that is not due to the lack of knowledge. The error can be categorized as either acknowledged or unacknowledged. Examples of acknowledged errors are round-off errors in a digital computer and physical approximations made to simplify the modelling of a physical process. Unacknowledged errors include blunders and mistakes such as programming errors.

In CFD simulations, there are four predominant sources of error, namely insufficient spatial discretization convergence, insufficient temporal discretization convergence, lack of iterative convergence, and computer programming. The most important activity in verification testing is to systematically refine the grid size and the time step. The objective of this activity is to estimate the discretization error of the numerical solution. As the grid size and time step approach zero, the discretization error should asymptotically approach zero. In verification activities, comparing a computational solution to a highly accurate solution is the most accurate and reliable way to quantitatively measure the error in the computational solution. However, highly accurate solutions are known for a relatively small number of simplified problems. These highly accurate solutions can be classified into three types: analytical solutions, benchmark numerical solutions to the ordinary differential, and benchmark numerical solutions to partial differential equations.

5.6 CFD CALCULATION

CFD is applied by first dividing or discretizing the geometry of interest into several computational cells. Discretization is the method of approximating the differential equations by a system of algebraic equations for the variables at some set of discrete locations in space and time. The discrete locations are referred to as the grid or the mesh.

The continuous information from the exact solution of the Navier-Stokes partial differential equations is now replaced with discrete values. The number of cells can vary from a few thousand for a simple problem to millions for very large and complicated ones. Cells have a variety of shapes. Triangular and quadrilateral cells are generally used in 2D problems. For 3D problems, hexahedral, tetrahedral, pyramidal, and prismatic-shaped cells can be used.

In the past, CFD codes required the use of structured grids containing one cell type, such as brick-shaped hexahedral elements, in which the cells were positioned in regular patterns. Current codes allow cells to be located in an irregular, unstructured pattern, giving much greater geometric flexibility. Additionally, a good CFD code can accept grids consisting of a combination of different cell types, or hybrid grids, to address complex geometries, providing flexibility to the CFD analyst. Geometries are often created using computer-aided design (CAD) software. The geometry, either a wire frame or solid model is exported to the grid-generation software program to create the CFD quality grid. A few packages have combined both functions of CAD geometry creation and mesh generation into a single interface. With the grid created, the boundary conditions such as pressures, velocities, mass flows, and scalars specified, and physical properties defined, the CFD calculations can start. The CFD codes will solve the appropriate conservation equations for all grid cells using an iterative procedure. Typical chemical process applications involve solving for mass conservation (using a continuity equation), momentum (using Navier Stokes equations), enthalpy, turbulent kinetic energy, turbulent energy dissipation rate, chemical species concentrations, and local reaction rates, and local volume fractions for multiphase problems.

There are many commercial CFD packages for modelling and analysing system involving fluid flow, heat transfer, and associated phenomena such as chemical reactions. Some popular CFD packages include FLUENT, CFX, PHOENICS, and ANSYS. All these commercial CFD codes contain three main elements: Pre-processor, Solver, and Post-processor. This study concentrates on the use of the ANSYS 2022 R1 software package to simulate the flow and mixing behaviour, especially for chemical and thermal industrial applications.

CHAPTER 6

COMPUTATIONAL DOMAIN

The first step in CFD analysis is to create a geometric model of the issue domain. The geometric model is created, and then it is meshed to discretize it. Specifying the input data, defining the material qualities, and defining the boundary conditions are done after meshing. The solution setup is then initialized, and computations are carried out for the designated number of iterations. After the calculations are finished, post processing is carried out to extract the data pertaining to the key variables. Since the desired results, contours, surface Nusselt number, is monitored and plotted during the calculation.

6.1 STEADY STATE ANALYSIS

The flow is assumed to be steady incompressible, so pressure-based solver is used for the numerical analysis. The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is used as the solution method. The flow is assumed to be at steady state, the fluid physical properties are constant and the effect of gravity is considered. For this numerical simulation flow is assumed to be incompressible, turbulent with constant fluid properties. The computational domain is shown below.

6.2 MODELLING

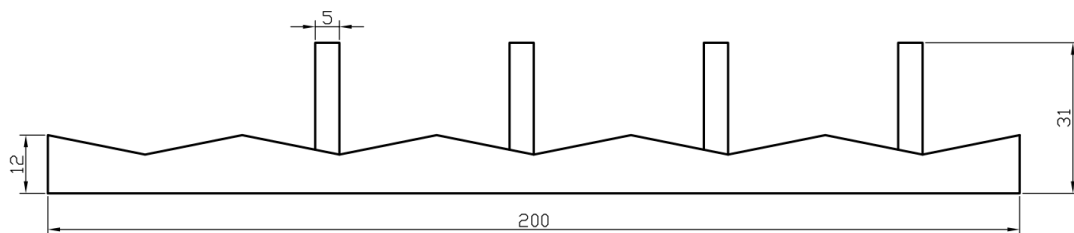


Fig 6.1 The dimensions of the BTMS

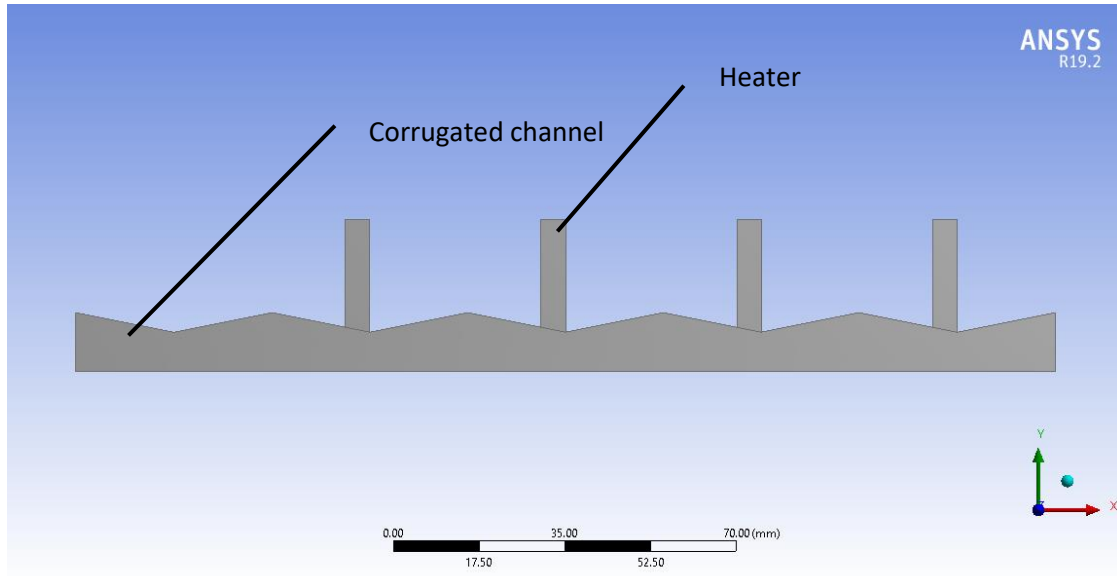


Fig 6.2 Numerical computational domain.

The two-dimensional model has been developed with the help of ANSYS 19.2 software. The two-dimensional numerical computational domain consists of an inlet, outlet, fluid domain and a heater. The heating Nichrome wire has a size of 5mm×23mm.

Table 6.1 Dimensions of Computational domain	
Geometrical details	Value
Number of heaters	4
Dimension of heater	5mm×23mm
Corrugated channel height	12 mm
Length of the corrugated channel	200 mm
Angle of bend	53.5°

6.3 MESHING

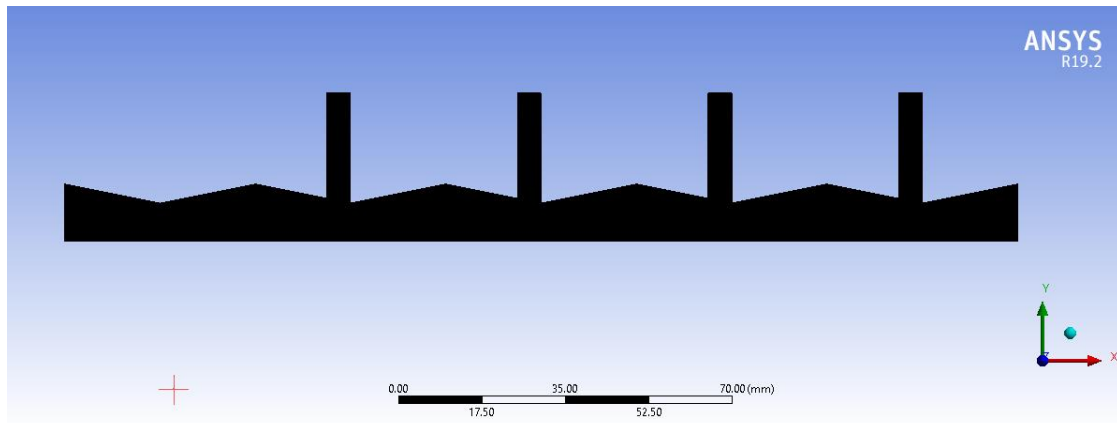


Fig 5.3 Mesh domain

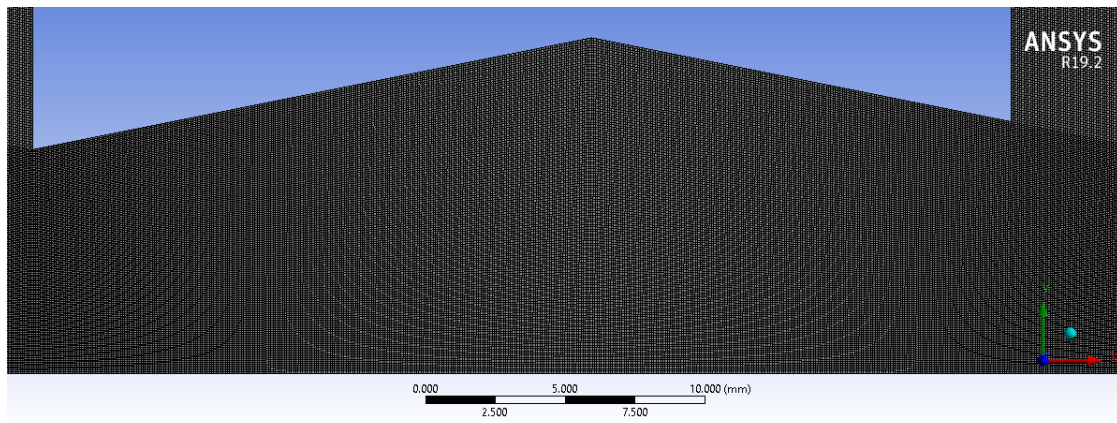


Fig 5.4 Fluid domain

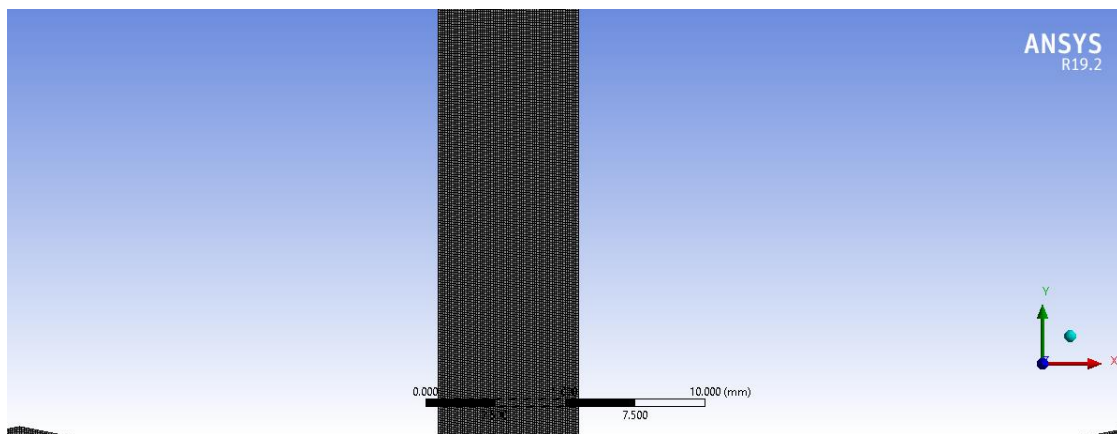


Fig 5.5 Heater domain

Computational domain contains around 422720 elements. Face sizing for inlet and set an appropriate value for relevance. Local inflation is done under the fluid domain

with appropriate number of inflation layers. All other settings are to be done as default. Once the mesh has been created, it must be checked for quality.

6.4 FLUENT ANALYSIS

In the present fluid domain, uniform velocity profile is assigned as the inlet boundary condition. The inlet fluid having temperature of 298K enters the domain with a velocity of 0.118m/s. The flow is taken as laminar. The fluid and the material properties are assumed to be constant.

6.4.1 Boundary Conditions

For an accurate solution to a fluid flow problem, it is crucial to define the appropriate boundary types and boundary conditions. While some boundary conditions are set by simulation software, the majority are defined by physical phenomena. The inlet fluid temperature is given as 298k and the inlet velocity is given as 0.118m/s. A constant net temperature of 333K is applied to the heater and all other wall surfaces are assumed to be adiabatic. For outlet mass flow rate of 0.00454kg/s is given.

6.4.2 Input Parameters

Certain parameter values, such as material qualities and beginning parameter values must be entered as input parameters into the CFD tool. The table contains the values for the input parameters.

Table 6.2 Input parameters		
Function	Specification	
Solver	Energy	On
	Type	Pressure-based
	Time	Transient
	Viscous model	Laminar
Materials	Solid	Nichrome
	Fluid	Water, Ethylene Glycol, EG-Water mixture of different ratio.

6.4.3 Convergence Criteria

The solution's convergence is determined by the difference in required parameter i.e., Nusselt number value after each iteration. Convergence criteria are established under the presumption that the solution would remain unchanged after convergence. One approach to the problem is the pressure-velocity coupling, which employs the SIMPLE system. The parameters were set in Fluent, and then the solution was initialized. Depending on how easily the convergence occurred and how long it took to obtain the results, iterations were performed.

CHAPTER 7

RESULTS AND DISCUSSION

In this chapter the numerical results are presented for the heat transfer characteristics of battery cooling through corrugated channel with water, ethylene glycol, ethylene glycol-water mixture. The material properties needed for the simulation is listed in Table 7.1.

The effect of coolant on the heat transfer enhancement of corrugated channel was studied. The thermal and fluid flow characteristics of the present problem were numerically simulated by computational fluid dynamics (CFD) technique. Commercial package ANSYS 19.2 is used for the present numerical study.

In this comparative study the effects of gravity is considered. No slip flow condition is assumed above the target surface.

Table 7.1 Material properties used in Numerical simulation.

	Nichrome	Water	EG	EG-Water (60:40)
Heat Capacity, C_p (J/kgK)	460.548	4182	2433	3216
Density, ρ (kg/m ³)	8200	998.2	113.2	1090
Thermal conductivity, K (W/mK)	13	0.6	0.254	0.375

7.1 MESH INDEPENDENT STUDY

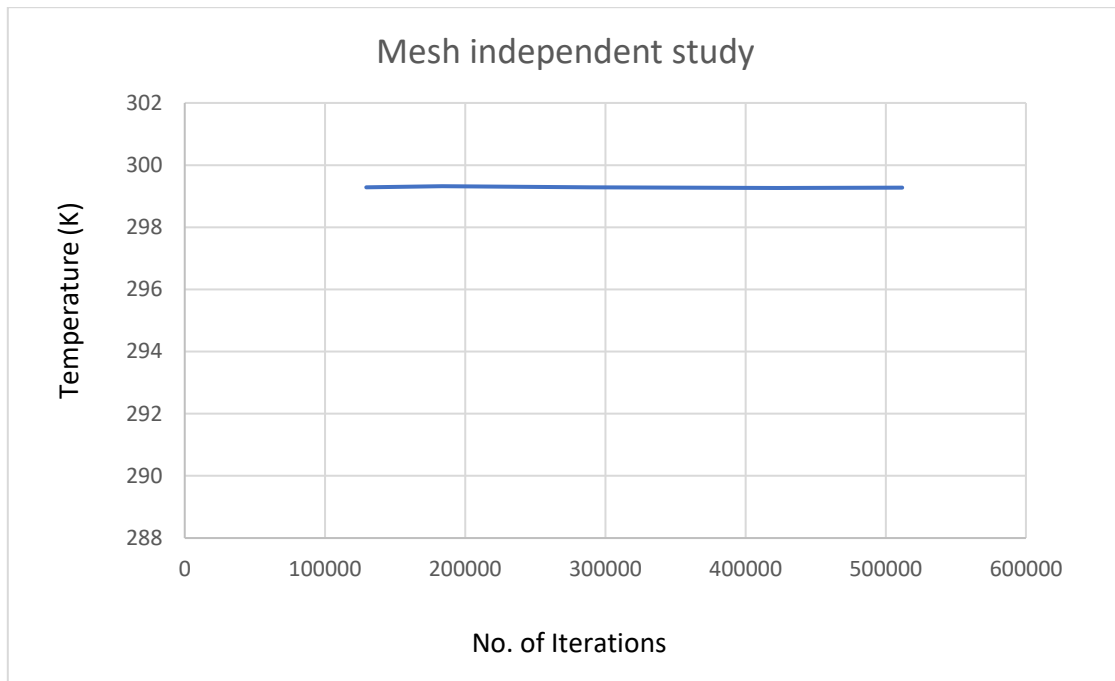


Fig 7.1 Variation of average Temperature with No. of elements

The average temperature shows much variation from the accurate value due to smaller No. of elements. The accuracy of the result increases with the element size and it reaches an optimum value, above which the variation is insignificant. Computational domain contains around 422720 elements.

7.2 VALIDATION OF NUMERICAL RESULTS

The corrugated channel was simulated for the normal operation of LIB. To assess the corrugated channel performance the input coolant flow velocity of 0.118m/s was examined for the simulation of 400s. The result for the maximum LIB surface temperature for 400s operation using water as coolant is shown in Fig 7.3. At the initial time the heater, corrugated channel, and coolant are at the equilibrium temperature of 60°C. The coolant is supplied at a constant temperature of 25°C. As seen, the maximum surface temperature of the LIB remains constant after about 300s and there is no significant change of temperature after that. It means the LIB, coolant and cooling plate reach temperature equilibrium in 200 s. Temperature contour of the simulation is shown in Fig 7.4 and flow vector is shown in Fig 7.5.

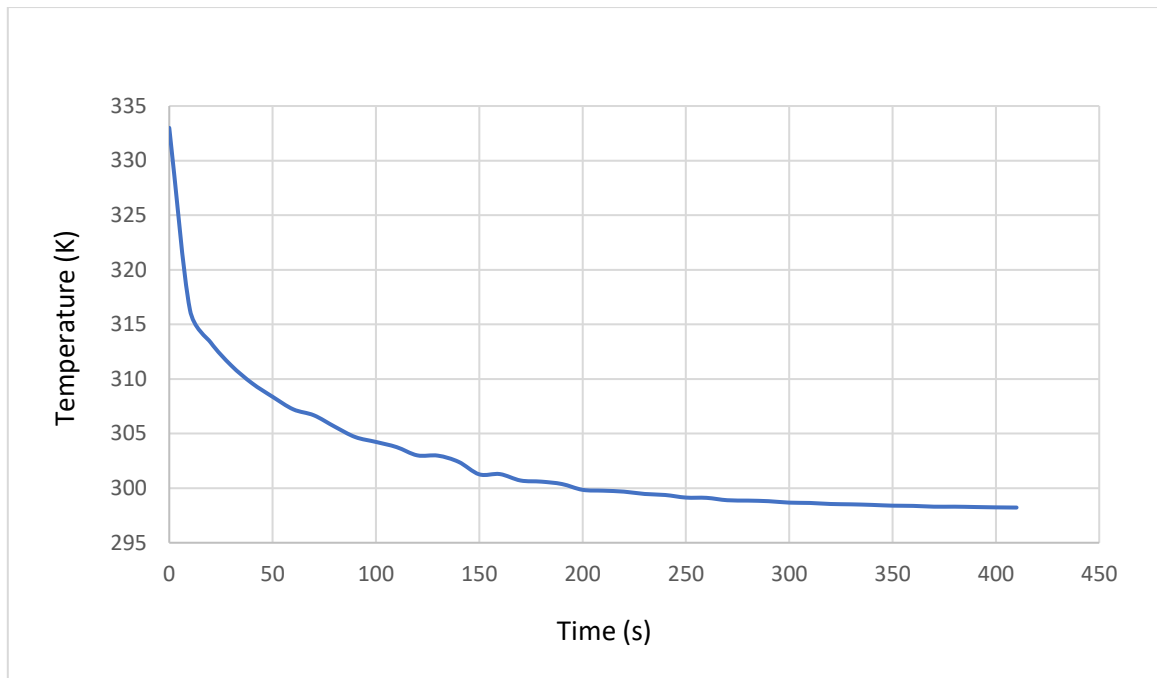


Fig 7.2 Maximum surface temperature of corrugated channel using water.

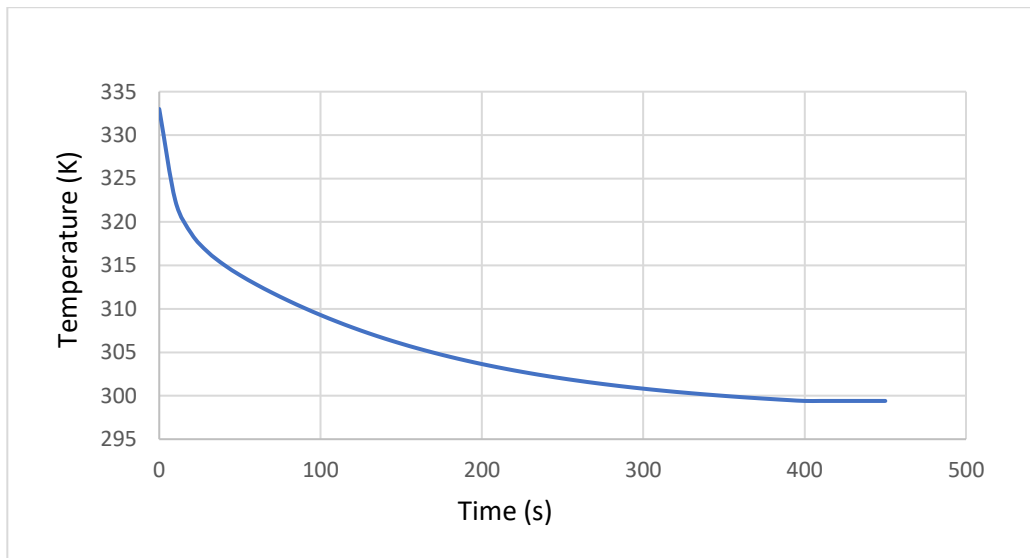


Fig 7.3 Maximum surface temperature of corrugated channel using ethylene glycol.

The result for the maximum LIB surface temperature for 450s operation using ethylene glycol as coolant is shown in Fig 7.6. All the operating condition for this simulation is same as the above. In this simulation equilibrium temperature reaches after

400s. After that the temperature remains constant. Temperature contour of the simulation is shown in Fig 7.7 and flow vector is shown in Fig 7.8.

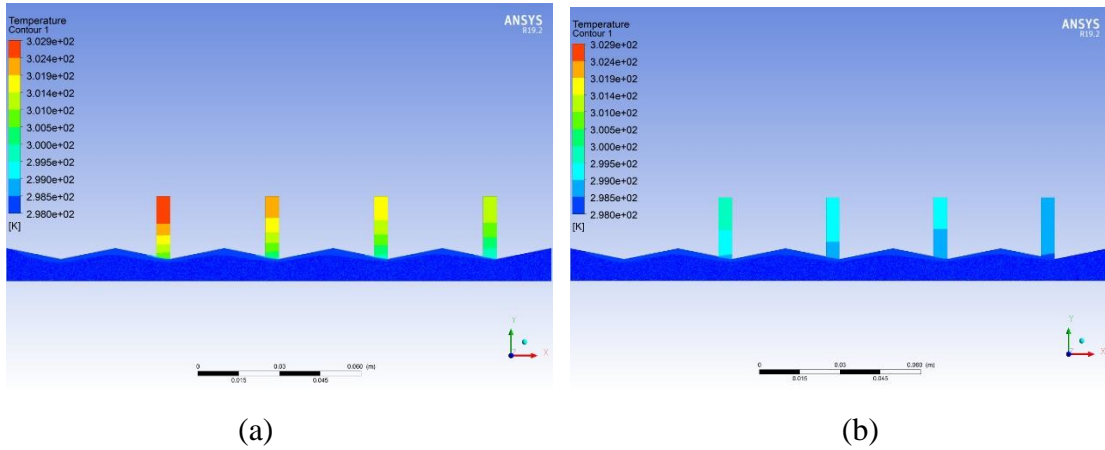


Fig 7.4 Temperature contour of corrugated channel using EG at (a).300s, (b). 450s.

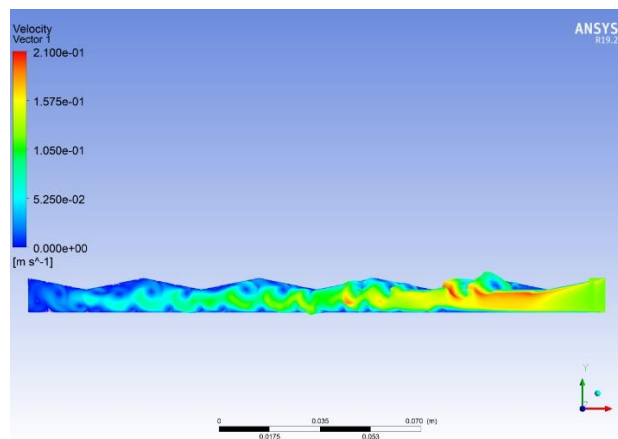


Fig 7.5 Velocity vector of corrugated channel using EG

The result for the maximum LIB surface temperature for 400s operation using ethylene glycol as coolant is shown in Fig 7.9. All the operating condition for this simulation is same as the above. In this simulation equilibrium temperature reaches after 360s. After that the temperature remains constant. Temperature contour of the simulation is shown in Fig 7.9 and flow vector is shown in Fig 7.10.

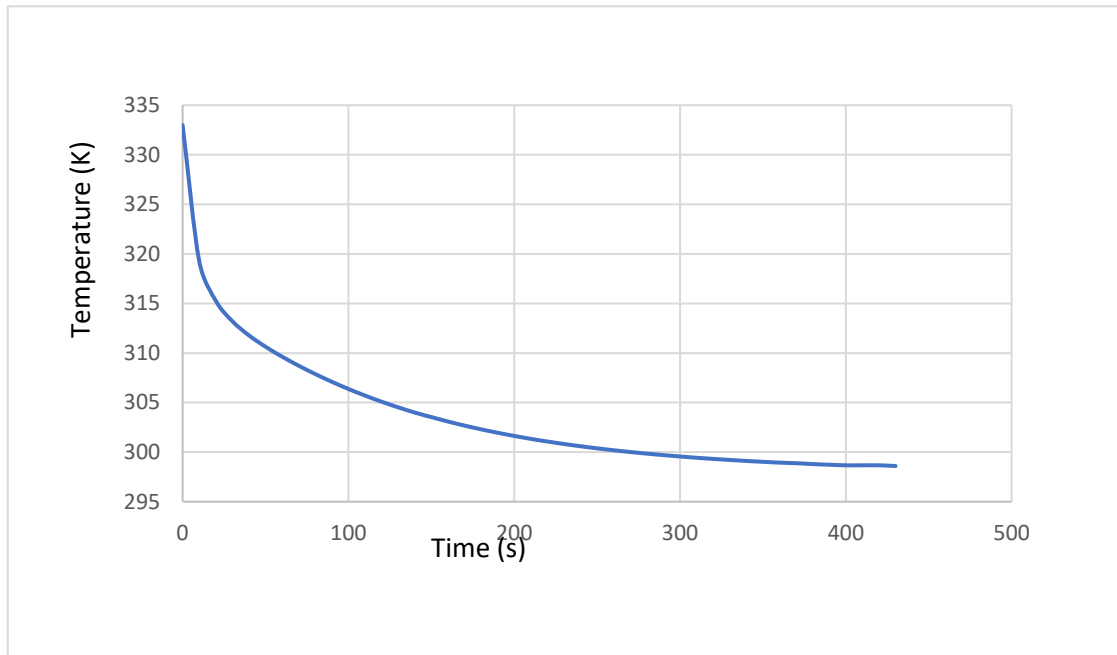


Fig 7.6 Maximum surface temperature of corrugated channel using EG-water (60:40).

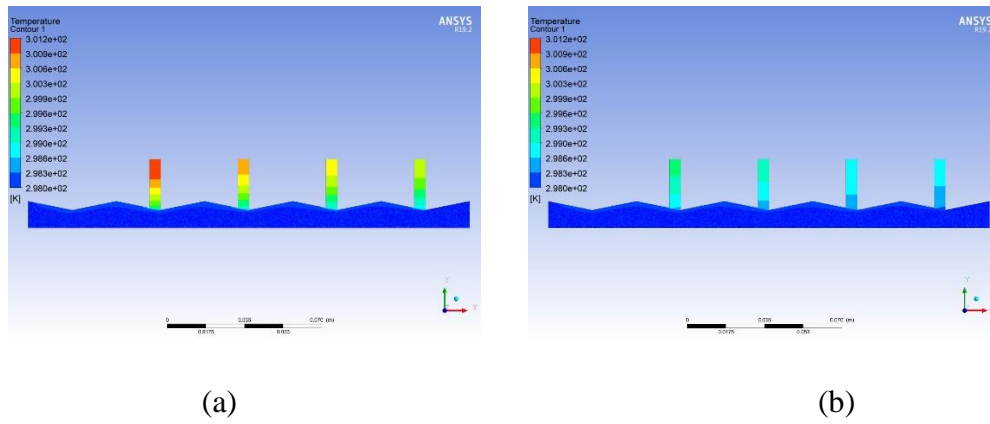


Fig 7.7 Temperature contour of corrugated channel using EG-Water (60:40) at (a).300s, (b). 400s.

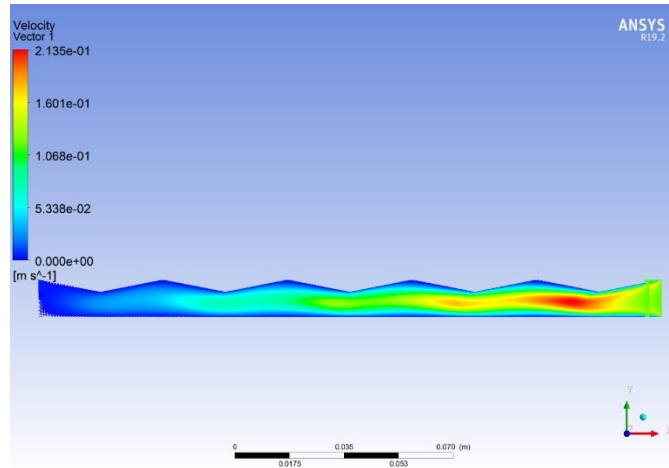


Fig 7.8 Velocity vector of corrugated channel using EG-Water (60:40).

7.3 SELECTION OF THE BEST COOLANT

The selection criteria for the cooling plate is the ability to maintain the LIB surface temperature below 27°C during normal operation. During normal operation all the coolant can maintain the battery temperature below 27°C with velocity of 0.118m/s.

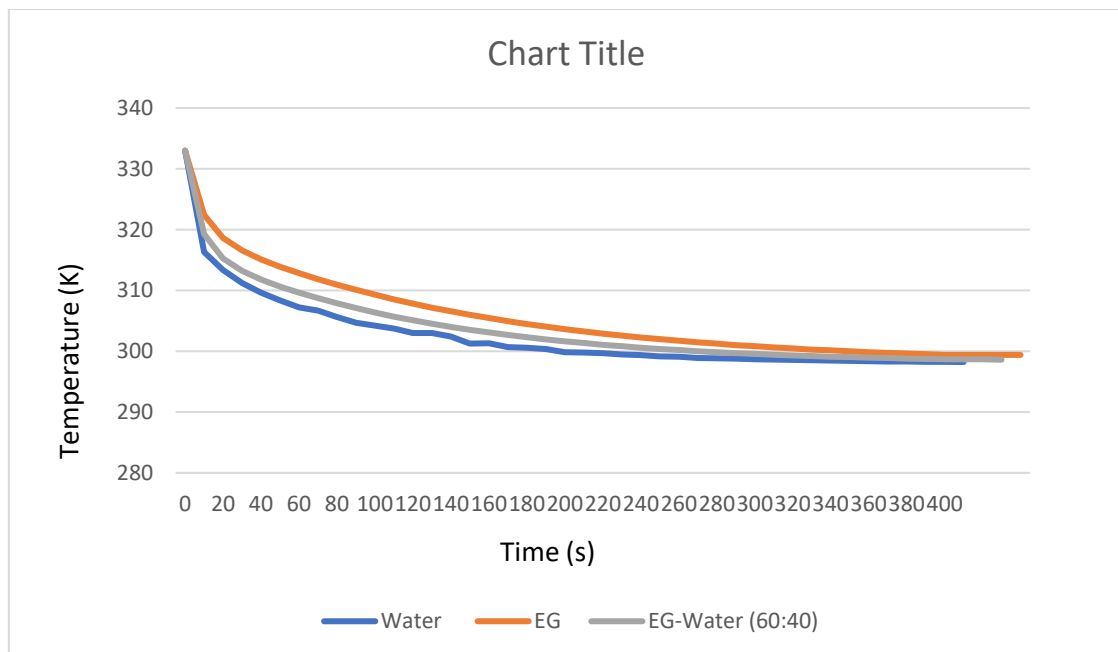


Fig 7.9 Comparison between different coolant performance.

7.4 COMPARISON BETWEEN EXPERIMENTAL DATA AND NUMERICAL DATA

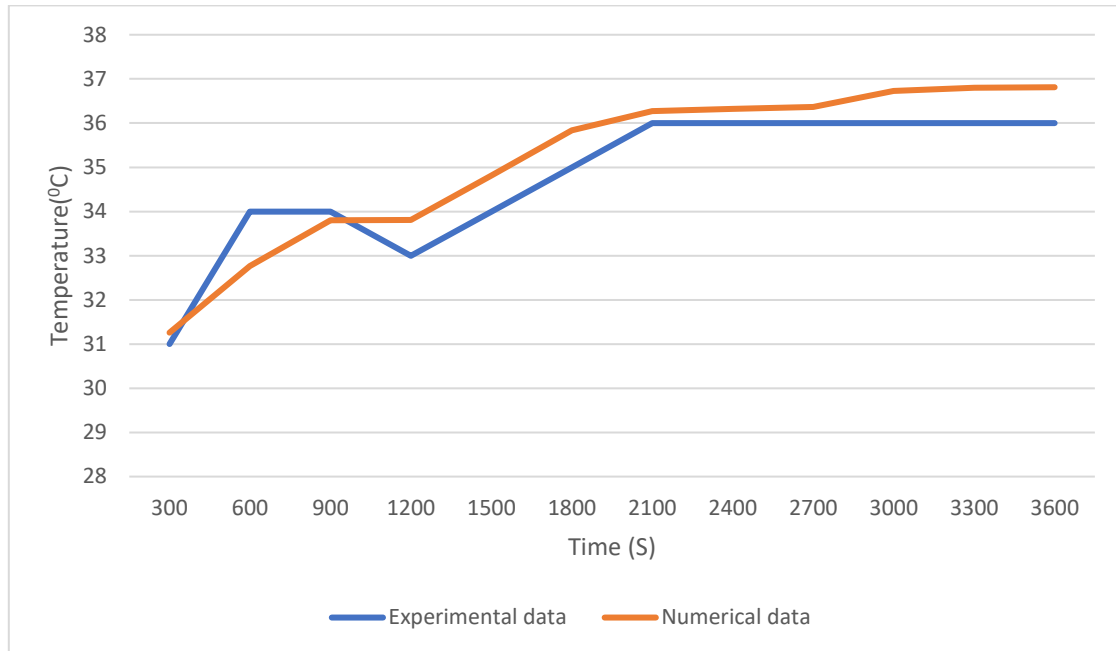


Fig 7.10 Comparison between experimental data vs numerical data.

Form the data of comparison between experimental vs numerical data up to 2100s temperature varies and beyond that temperature becomes almost similar and after 3000s temperature is similar.

CHAPTER 8

CONCLUSION

The novel design of BTMS was designed for thermal management of LIB packs to supply excessive cooling to combat the heat generation during normal operation and thermal runaway of the battery. This BTMS has featured with corrugated channel to extend the effective heat transfer area between the cooling plate and the coolant with minimum coolant pressure drop in the cooling plate.

A computer model was developed in ANSYS 19.2 to simulate the performance of the designed cooling plate. The corrugated channel was simulated based on the heat generation trend of aggressive discharging of the battery during the normal operation.

From the comparison between experimental and numerical data it can be seen that temperature distribution is almost similar. From the comparison between numerical simulation of different coolant, it can be clear that Ethylene glycol with mixture of water in the ratio of 60:40 having more heat transfer capacity than others. So, from the data EG-water (60:40) selected for BTMS for Li-Ion batteries.

REFERENCES

- ANSYS Fluent software package: user's manual 2022 R1, (2022).
- Abdul Haq Mohammed, Roja Esmaeeli, Haniph Aliniagerdroudbari, Muapper Alhadri, Seyed Reza Hashemi, Gopal Nadkarni, Siamak Farhad, (2019), Dual-purpose cooling plate for thermal management of prismatic lithium-ion batteries during normal operation and thermal runaway, *Applied thermal engineering* 1359-4311.
- Dattatraya G. Subhedara, Bharat M. Ramanib, Akhilesh Gupta, (2017), Experimental investigation of heat transfer potential of Al₂O₃/ Water-Mono Ethylene Glycol nanofluids as a car radiator coolant, *Thermal engineering* 2214-157X.
- Charlotte Roe, Xuning Feng, Gavin White, Ruihe Li, Huaibin Wang, Xinyu Rui, Cheng Li, Feng Zhang, Volker Null, Michael Parkes, Yatish Patel, Yan Wang, Hewu Wang, Minggao Ouyang, Gregory Offer, Billy Wu, (2022), Immersion cooling for lithium-ion batteries – A review, *Power sources* 0378-7753.
- Chaitanya Pandya, Dhrunil Timbadia², (2021), A Detailed Review on Cooling System in Electric Vehicles, *IRJET*.
- R. Prasanna Shankara, N.R. Banapurmath, Abhinandan D'Souza, A.M. Sajjan, N.H. Ayachit, T.M. Yunus Khan, Irfan Anjum Badruddin, Sarfaraz Kamangar, (2021), *AJE* 1110-0168.
- Winifred Nduku Mutuku, (2016), Ethylene glycol (EG)-based nanofluids as a coolant for automotive radiator, *Asia Pacific Journal on Computational Engineering* 1186/s40540-016-0017-3.
- Satish G. Kandlikar and Murat Bulut, (2003), An Experimental Investigation On Flow Boiling Of Ethylene-Glycol/Water Mixture, *Journal of Heat Transfer* 1115/1.1561816.