SEMARAK ILMU PUBLISHING

Semarak International Journal of Material Research

Journal homepage:
https://semarakilmu.my/index.php/sijmr/index
ISSN: 3083-8908




# Numerical Analysis of Transient Thermal Temperature Distribution in a Thermal Chamber: A Comparative Study of Heating and Cooling Conditions for Energy Component Testing

Yusuf Tri Hadi Mulyana[1], Rizky Samudera Febrian Putra[1], Iqbal Dwi Prayoga[1], Moouzzi Akbar Ramadan[1], Arya Kusumawardana[1,*]

[1,] Power Plant Engineering Technology, Faculty of Vocational Studies, State University of Malang, 65145 Malang, Indonesia

**ARTICLE INFO**

***Article history:***
Received 13 June 2026
Received in revised form 30 August 2026
Accepted 30 September 2026
Available online 9 October 2026

The demand for thermal testing systems with high accuracy and stability is increasing, particularly for evaluating the performance of energy components such as electric vehicle batteries and materials under extreme temperature conditions. However, conventional thermal chambers still face thermal non-uniformity caused by complex heat transfer mechanisms, resulting in large temperature gradients and reduced testing validity. This study aims to numerically analyze temperature distribution within a thermal chamber using a transient thermal simulation approach based on the finite element method (FEM) by evaluating heating and cooling characteristics over time in a closed system. The analysis considered heat transfer mechanisms including conduction through chamber walls, convection with the surrounding environment, and boundary conditions in the form of heat flux. Heating and cooling processes were simulated to evaluate temperature distribution, thermal gradients, and thermal stability. The results show that during heating for 600 s, the average temperature increased from 16.09°C to 44.86°C, with a maximum temperature of approximately 65°C and a minimum temperature of about 22°C, indicating non-uniform heat distribution. During cooling, the average temperature decreased from 6.49°C to 1.41°C, with a minimum temperature of −11.76°C while the maximum temperature remained around 21.6°C, revealing significant thermal deviation within the chamber. Overall, the results demonstrate that temperature distribution is strongly influenced by heat transfer mechanisms and boundary conditions. Thermal non-uniformity remains a critical factor affecting testing accuracy, highlighting the need for design optimization to improve thermal homogeneity, system stability, and the reliability of thermal chambers for modern energy component testing applications.

***Keywords:***
Transient thermal; thermal chamber; temperature distribution; finite element method; thermal non-uniformity; energy components

## 1. Introduction

This study was conducted using a numerical simulation approach based on the finite element method (FEM) to analyze the transient temperature distribution in a thermal chamber system used

[*] *Corresponding author.*
*E-mail address: arya.kusumawardana.ft@um.ac.id*

for testing energy components [1]. The simulation was performed to evaluate the system's thermal performance under two main conditions—heating and cooling—while considering temperature changes over time within a closed system [2]. The thermal chamber geometry is modeled as a test chamber with solid walls undergoing simultaneous heat transfer interactions, including conduction through the wall material, convection with the surrounding environment, and the application of boundary conditions in the form of heat flux on specific surfaces. The initial system temperature conditions are determined based on simulation data, with different initial temperature ranges for each heating and cooling scenario [3,4].

In this simulation, the material's thermophysical parameters—such as thermal conductivity, specific heat capacity, and density—are defined according to the characteristics of the materials used to construct the chamber [5,6]. Additionally, boundary conditions, including ambient temperature and the convective heat transfer coefficient, are set to represent real-world operating conditions [7,8]. The heating process is analyzed by gradually applying heat input until the simulation time reaches 600 seconds, while the cooling process is modeled by heat dissipation from the system to the environment [9,10]. Temperature distributions are observed at several measurement points to obtain minimum, maximum, and average temperature values as indicators of system performance [11,12].

The energy conservation equation was solved numerically in a transient form to capture the dynamics of temperature changes over time [13,14]. The simulation was performed iteratively until solution convergence was achieved, while monitoring residual stability at each time step [15,16]. The results obtained—spatial and temporal temperature distributions—were then analyzed to evaluate thermal non-uniformity, temperature gradients, and the system's ability to reach a steady-state thermal condition [17,18].

Analysis of the results shows that under heating conditions, there was a significant increase in the average temperature, but this was accompanied by a considerable temperature difference between the maximum and minimum points [19,20]. This indicates an uneven heat distribution within the chamber [21,22]. Meanwhile, under cooling conditions, the temperature decrease was uneven, with extreme differences between the minimum and maximum temperatures, indicating that heat loss was concentrated in certain areas, particularly near the walls [23].

This non-homogeneous temperature distribution is a major concern because it can affect the validity of testing energy components that require stable and uniform thermal conditions [24,25]. Therefore, the results of this simulation are used to evaluate the heat transfer characteristics within the system and serve as a basis for developing a more optimal thermal chamber design. This analysis is expected to contribute to improving the efficiency, stability, and reliability of thermal testing systems in modern energy applications [26].

**Table 1**
State of the art

| References | Method | Material / System | Key Results |
|---|---|---|---|
| [27] | FEM numerical simulation (transient thermal) | Thermal chamber (enclosed space + solid walls) | Temperature distribution is non-uniform, influenced by conduction mechanisms in the walls and *heat flux* as the heat source |
| [28] | CFD (ANSYS Fluent), convection analysis | Thermal test chamber with airflow | Airflow patterns affect temperature distribution; stagnant zones cause high *thermal gradients* |

**Table 1 (*Continued*)**

| | | | |
|---|---|---|---|
| [29] | Transient heat transfer simulation | Electric vehicle battery system | Temperature inhomogeneity accelerates battery performance degradation; effective thermal management is required |
| [30] | FEM + conduction-convection analysis | Composite materials in extreme temperature environments | Heat distribution is strongly influenced by material thermal properties and environmental boundary conditions |
| [31] | CFD + thermal radiation | Thermal enclosure (enclosed space with radiant heating) | Radiation accelerates the rise in temperature but causes uneven heat distribution |

Previous numerical simulation studies employing transient FEM and CFD approaches have demonstrated sufficient accuracy in analyzing temperature distributions within thermal chamber and thermal enclosure systems, particularly in identifying the effects of conduction, convection, and radiation mechanisms on heat distribution patterns [32]. However, most existing studies address only partial aspects of heat transfer—such as fluid flow dominance or material-specific effects—without comprehensively examining the time-dependent dynamics of temperature change within a closed system [33]. Notably, Biçer *et al*., [3] focused on battery chemistry and assembly but did not address thermal uniformity in the testing environment, while Chaffer *et al*., [4] designed an atmosphere-controlled furnace without evaluating convective heat distribution under transient conditions. Studies such as those by Lyu *et al*., [7] and Hussien *et al*., [14] focused on battery pack-level thermal management rather than chamber-level performance characterization. Furthermore, research that simultaneously and comparatively evaluates both heating and cooling processes within the same thermal chamber model under transient conditions remains very limited, particularly in the context of quantifying thermal non-uniformity and thermal gradient as key performance metrics [34].

This research gap indicates the need for a comprehensive transient thermal simulation model that can analyze both spatial and temporal temperature distributions within a thermal chamber across both heating and cooling conditions. Therefore, this study aims to: (1) numerically analyze transient temperature distribution in a solid-state thermal chamber system under heating and cooling scenarios using FEM, and (2) comparatively evaluate the performance of natural convection versus forced convection in achieving thermal uniformity and system stability. The significance of this study lies in its contribution to the development of more reliable and thermally homogeneous chamber designs for energy component testing, particularly for electric vehicle battery validation where temperature uniformity is critical to the validity and reproducibility of test results.

## 2. Methodology

### *2.1 Research Flowchart*

This study employs a numerical simulation approach based on Transient Thermal Analysis to analyze temperature distribution within a thermal chamber system under time-dependent conditions, taking into account the interaction between heating and cooling processes within an enclosed system. The simulation was performed using the Finite Element Method (FEM), accounting for heat transfer mechanisms such as conduction through the walls, convection with the environment, and the influence of boundary conditions involving heat flux as both a heat source and

a heat sink. During the heating phase, the system receives thermal energy from an internal source, causing the temperature to rise gradually; conversely, during the cooling phase, the system releases heat to the environment, causing the temperature to decrease over time.

The physical model is assumed to be three-dimensional, with a domain consisting of a closed space representing a thermal chamber for testing energy components. Heat transfer within the system is dominated by conduction through solid materials and natural convection between the internal surfaces and the air inside the chamber. During the heating process, heat distribution is influenced by the location of the heat source and the thermal properties of the material, whereas during the cooling process, temperature distribution is influenced by heat loss mechanisms through the walls and interactions with the surrounding environment. These interactions result in temperature gradients that vary spatially and temporally within the system.

Numerical validation was performed by evaluating the consistency of the simulation results' trends with expected physical phenomena, such as the gradual increase in temperature during the heating process and the decrease in temperature during the cooling process. Additionally, validation was assessed based on the formation of thermal gradients and the difference between maximum and minimum temperatures as indicators of thermal non-uniformity. Under heating conditions, validation is demonstrated by an increase in the average temperature until it approaches a steady state, whereas under cooling conditions, it is demonstrated by a non-uniform temperature decrease due to the dominance of heat loss in certain areas [36].

Heat entering or leaving the system is then distributed through three main mechanisms: conduction through solid materials such as the chamber walls, convection between the surface and the air inside the chamber, and thermal interaction with the external environment [37]. During heating, heat spreads from the source throughout the entire domain, whereas during cooling, heat is released into the environment, resulting in a gradual decrease in temperature [38].

The thermal chamber system is designed as an enclosed space with wall materials possessing specific thermal insulation properties to control the rate of heat transfer [39]. This design aims to simulate testing conditions for energy components that require temperature stability. However, limitations in heat distribution within the system result in the formation of significant temperature gradients. The main components of the system consist of a heat source (heat flux), a test chamber (thermal chamber), wall materials acting as a conduction medium, and the external environment acting as a heat dissipation medium [40]. The interaction between these components forms a complex thermal system, where system performance is greatly influenced by temperature distribution and the system's ability to achieve uniform and stable thermal conditions. This is illustrated in Figure 1, a flowchart.

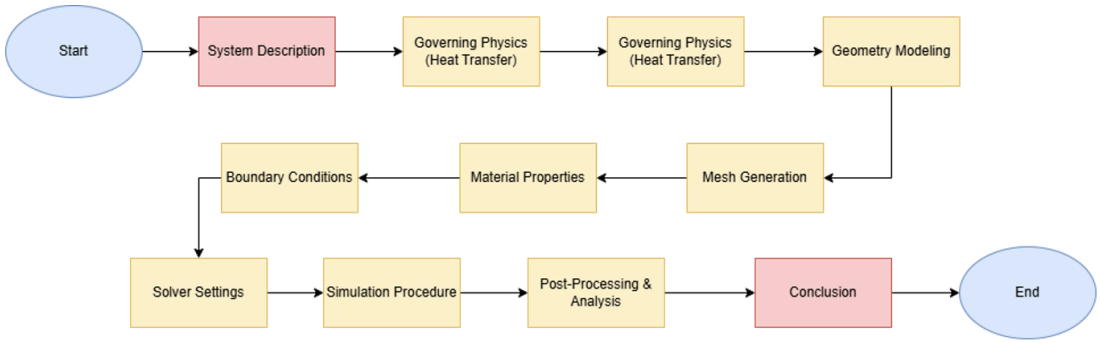



**Fig. 1.** Flowchart

### *2.2 System Description*

The system analyzed is a thermal chamber system that operates in a closed-loop configuration (enclosure system) utilizing three main components: a fan, a Positive Temperature Coefficient (PTC) heating element, and a thermoelectric (Peltier) module to generate controlled heating and cooling conditions. Electricity is used to operate the PTC element as a heat source during heating, and the Peltier module as a cooling component capable of absorbing heat from inside the chamber and releasing it to the external environment. The generated heat and cold are then distributed throughout the test chamber via heat transfer mechanisms such as conduction through the walls, air convection within the chamber, and thermal interaction with the surrounding environment. In fanless conditions, temperature distribution within the chamber is dominated by natural convection due to differences in air density, whereas in fan-assisted conditions, forced airflow is used to enhance circulation and temperature homogeneity. The system is designed as a sealed chamber to minimize external disturbances and maintain thermal stability during the testing process. The main components of the system include:

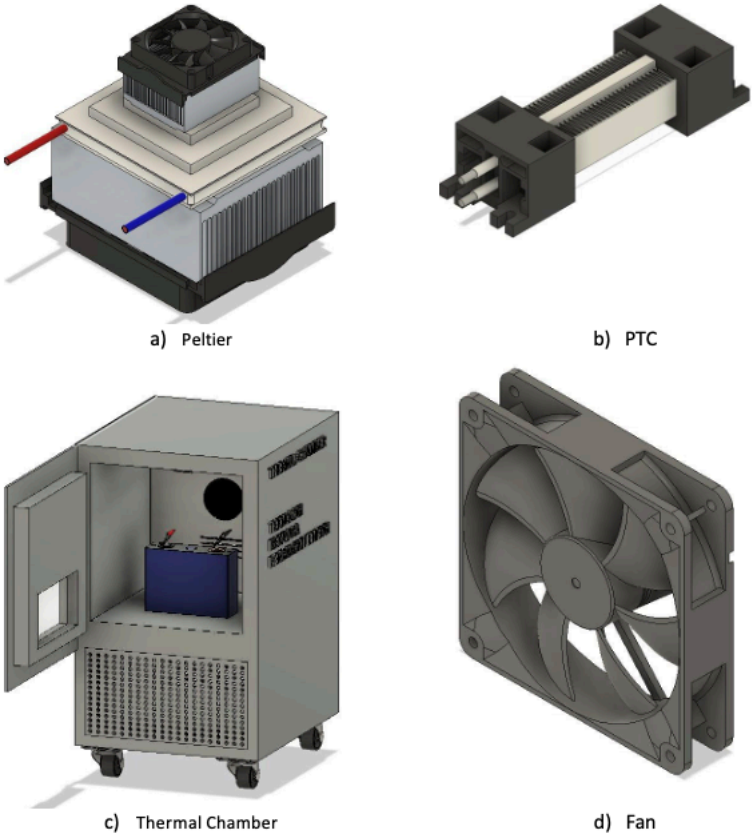



**Fig. 2.** Main components

Operationally, electrical energy is used as the primary source to activate the heating and cooling components within the thermal chamber system, namely the PTC elements and Peltier modules. The electrical energy supplied to the PTC elements is converted into thermal energy through the Joule effect, thereby increasing the temperature inside the chamber during the heating phase. Meanwhile, the Peltier module operates based on the thermoelectric effect, where one side of the module absorbs heat (the cold side) from inside the chamber and the other side releases heat to the external environment. This allows the system to generate two distinct thermal conditions within a single

configuration: heating and cooling. The self-regulating nature of PTC allows the system to maintain temperature stability without requiring complex controls, while the Peltier module provides flexibility in the controlled cooling process. The combination of these two components enables the thermal chamber system to simulate extreme temperature conditions more adaptively and efficiently.

The heat and cold generated by the PTC elements and Peltier modules are then distributed throughout the test chamber via three primary heat transfer mechanisms: conduction, convection, and thermal radiation. Conduction occurs through solid materials such as the chamber walls and component surfaces, while convection results from the interaction between surfaces and the air inside the chamber. Under fanless conditions, heat transfer is dominated by natural convection due to differences in air density triggered by temperature gradients, resulting in natural flow patterns that tend to produce thermal stratification. Conversely, under fan-assisted conditions, airflow is forced to enhance circulation, thereby accelerating the distribution of thermal energy and improving temperature homogeneity within the chamber. Additionally, thermal radiation also plays a role in directly transferring energy between surfaces with different temperatures, particularly under high-temperature heating conditions.

The thermal chamber system is designed as an enclosed system to ensure that the test environment remains controlled and stable. The chamber walls are constructed from materials with specific thermal insulation properties to reduce the rate of heat transfer to the environment, whether during heating or cooling. During the heating process, thermal insulation helps retain heat energy within the system, while during the cooling process, it helps control the rate at which heat enters from the external environment. This design allows the system to achieve the desired temperature conditions more efficiently, although there is still the potential for temperature gradients to form due to uneven energy distribution within the chamber.

The main components of the system consist of a PTC element as the heat source, a Peltier module as the cooling component, a fan to drive airflow and enhance forced convection, a thermal chamber as the primary domain of analysis, and an insulating wall to inhibit heat exchange with the environment. The interaction between these components forms a complex integrated thermal system, where overall performance is significantly influenced by temperature distribution, heat transfer efficiency, and the system's ability to achieve uniform thermal conditions during both heating and cooling processes.

### *2.3 Governing Physics*

Heat transfer phenomena within a thermal chamber are governed by conduction and convection mechanisms under transient conditions, where temperature varies with time. To accurately represent these phenomena, a numerical approach based on the finite element method is employed to solve the energy conservation equations in their transient form.

$$\rho c_p \frac{\partial T}{\partial t} = k\nabla 2T + Q \quad (1)$$

This equation shows that the change in temperature over time is influenced by the material's thermal conductivity, heat capacity, and internal heat source (Q). In the context of a thermal chamber, the value of Q represents heat input during heating and heat dissipation during cooling. In addition to conduction, heat transfer also involves convection mechanisms on surfaces interacting with a fluid (air), as described by Newton's law of cooling.

$$q = hA(Ts - T\infty) \quad (2)$$

This equation shows that the rate of convective heat transfer depends on the heat transfer coefficient (h), the surface area (A), and the temperature difference between the surface and the surrounding environment. In this simulation, the system is analyzed under transient conditions, so temperature changes are observed as a function of time. This approach allows for the evaluation of the system's dynamics as it reaches a steady state, whether during heating or cooling.

### *2.4 Surveyors and Mesh*

The meshing process is a crucial step in Computational Fluid Dynamics (CFD) simulations because it directly affects the accuracy and stability of the numerical solution. In this study, the computational domain created during the geometric modeling stage was discretized into small elements using the ANSYS Meshing module. This discretization enables the numerical solution of the mass, momentum, and energy conservation equations for each control volume within the system, under both natural convection and forced convection conditions.

The mesh type used is an unstructured tetrahedral mesh, selected for its ability to handle the complex geometry of the thermal chamber system—specifically the enclosure—in the areas surrounding the PTC element as the heat source, the Peltier module as the cooling component, the fan as the airflow driver, and the corners of the enclosure, which have the potential to cause high temperature gradients. Additionally, to improve accuracy in capturing heat transfer phenomena near the walls, an inflation layer feature is applied to the solid-fluid interface. This layer enables better resolution of temperature gradients in the boundary layer region, which is a critical area in analyzing temperature distribution, both under heating and cooling conditions within the thermal chamber.

To ensure good mesh quality, several evaluation parameters are used, including keeping the skewness value below 0.85 (ideally close to 0), ensuring the orthogonality quality is greater than 0.2, and maintaining the aspect ratio within reasonable limits to avoid element distortion. These parameters are crucial for ensuring the stability of the numerical solution and the accuracy of simulation results, particularly when representing transient thermal phenomena involving temperature changes over time.

In addition, local mesh refinement was performed in areas with high temperature gradients, particularly around the PTC elements, the surface of the Peltier module, and the hot and cold distribution channels within the chamber. This was done to improve the numerical resolution in areas experiencing significant temperature changes—both during heating and cooling—so that the temperature distribution could be captured in greater detail.

To ensure that the simulation results are not dependent on the mesh size, a grid independence test was conducted. In this test, simulations were performed with several variations in the number of elements (e.g., 100,000, 250,000, and 500,000 elements), and key parameters such as maximum and minimum temperatures, as well as the average temperature distribution during heating and cooling conditions, were compared. The mesh is considered independent if the difference in results between variations is less than 5%. This approach ensures that the simulation results have a high level of reliability in representing transient temperature distribution phenomena in the thermal chamber system. For details on the thermal chamber used for the simulation, please refer to the following figure 3.

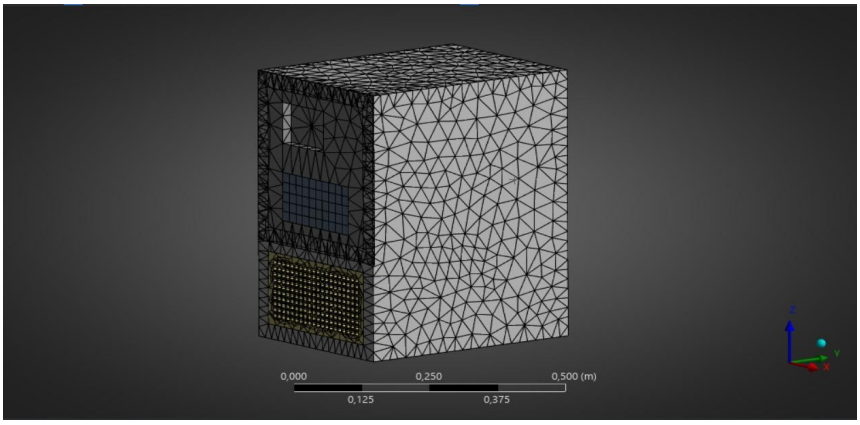



**Fig. 3**. Mesh box

### *2.5 Material Properties*

In this simulation, the materials used consist of air as the working fluid within the thermal chamber, as well as solid materials that form the system walls, PTC heating elements, and Peltier cooling modules. Air was chosen as the primary medium because it plays a direct role in the temperature distribution process through both natural convection and forced convection mechanisms within a closed space. Under fanless conditions, air facilitates natural circulation due to density differences triggered by temperature gradients, whereas under fan-assisted conditions, air serves as the primary medium for heat transfer via forced flow, which enhances temperature homogeneity within the chamber. Meanwhile, solid materials such as the enclosure walls, PTC elements, and Peltier components were selected based on their thermal conductivity characteristics, which influence the rate of heat transfer and thermal energy distribution within the system. The combination of these material properties is critical in analyzing temperature distribution, thermal gradients, and heat transfer efficiency in the thermal chamber system under both heating and cooling conditions. The material parameter values used in the simulation, including physical properties such as density, thermal conductivity, specific heat capacity, and dynamic viscosity, are presented in Table 3.

**Table 3**
Material properties

| No | Material | Density ($kg/m^3$) | Thermal Conductivity ($W/m\cdot°C$) | Heat Capacity ($J/kg\cdot°C$) | Dynamic Viscosity ($Pa\cdot s$) |
|---|---|---|---|---|---|
| 1 | Air (Water) | 1.225 | 0.024 | 1006 | $1.789 \times 10^{-5}$ |
| 2 | Steel | 7,850 | 60.5 | 434 | - |
| 3 | Wood, Oak | 935.7 | 0.45 | 1685 | |
| 4 | PTC Copper | 8300 | 401 | 385 | - |

The properties of this material significantly affect heat distribution, where materials with high thermal conductivity, such as copper, accelerate heat transfer, while insulating materials, such as wood, slow down heat transfer.

### *2.6 Boundary Conditions*

The boundary conditions used in this simulation were defined to represent the actual operating conditions of the thermal chamber system in a closed environment, taking into account the heat input from the PTC element, the cooling effect of the Peltier module, thermal interaction with the environment, as well as natural convection due to density differences and forced convection caused by airflow from the fan. In fanless conditions, the temperature distribution inside the chamber is entirely controlled by natural convection triggered by the temperature gradient, whereas in conditions with a fan, the influence of forced flow is added, which increases air circulation, thereby accelerating the distribution of heat and cold and improving temperature homogeneity within the system.

**Table 4**
Boundary conditions in the simulation

| No | Component | Boundary Condition Type | Parameter | Value / Description |
|---|---|---|---|---|
| 1 | PTC Heater | Heat Source /Heat Flux | Heat Output (Q) | 100 W |
| 2 | Enclosure Wall | Adiabatic /Convection | Convection coefficient (h) | 5–10 W/m²K (if there is heat loss) |
| 3 | Peltier | Heat Sink /Cooling Flux | Cooling rate (Q) | Variable (cooling conditions) |
| 4 | Module Fan | Inlet Velocity/Forced Convection | Flow velocity (v) | Variable (active during forced convection) |
| 5 | Ambient | Ambient Temperature | Temperature (T) | 22 °C |
| 6 | Temperature Gravity | Body Force | Acceleration due to gravity (g) | 9.81 m/s² |

The boundary conditions specified in Table 2 were established to realistically represent the operational conditions of the thermal chamber in two main scenarios: the heating and cooling processes within a closed system, where the PTC element is modeled as a heat source (heat flux) with a specific power to increase the temperature, while the Peltier module is modeled as a heat sink to lower the temperature inside the chamber. Additionally, the fan is modeled as a velocity inlet that generates forced convection with variable velocity to evaluate its effect on temperature distribution and system homogeneity; thus, in fanless conditions, heat transfer is dominated by natural convection, whereas in conditions with a fan, increased fluid mixing accelerates thermal energy distribution. The enclosure walls are modeled with convective or adiabatic boundary conditions using specific heat transfer coefficients ( ) to represent thermal interaction with the environment, while ambient temperature is assumed constant as an external boundary condition. The effect of gravity is also included as a body force to capture the natural convection phenomenon resulting from differences in air density, and time parameters are used in transient simulations to analyze changes in temperature distribution over time until approaching steady-state conditions. Thus, this combination of boundary conditions enables a comprehensive analysis of temperature distribution, thermal gradients, and thermal non-uniformity within the thermal chamber system.

## 3. Result

### *3.1 Heat Distribution Analysis*

#### *A. Conditions Without a Fan*

In the fanless condition, the temperature distribution inside the thermal chamber indicates that natural convection—driven by differences in fluid density—is the dominant mechanism. At the initial time (0 s), heat from the PTC heater element begins to spread by conduction to the surrounding air. Over time (150 s to 300 s), the heated air moves upward as its density decreases, while the cooler air sinks to the bottom. This phenomenon causes the formation of a significant vertical temperature gradient, where the upper part of the chamber tends to have a higher temperature compared to the lower part. Consequently, heat distribution becomes uneven and has the potential to create hotspots in certain areas. The simulation results can be seen more clearly in the following figure 4.

a) Conditions at 0 s
b) Conditions at 150 s
c) Conditions at 300 s
d) Conditions at 450 s
e) Conditions at 600 s

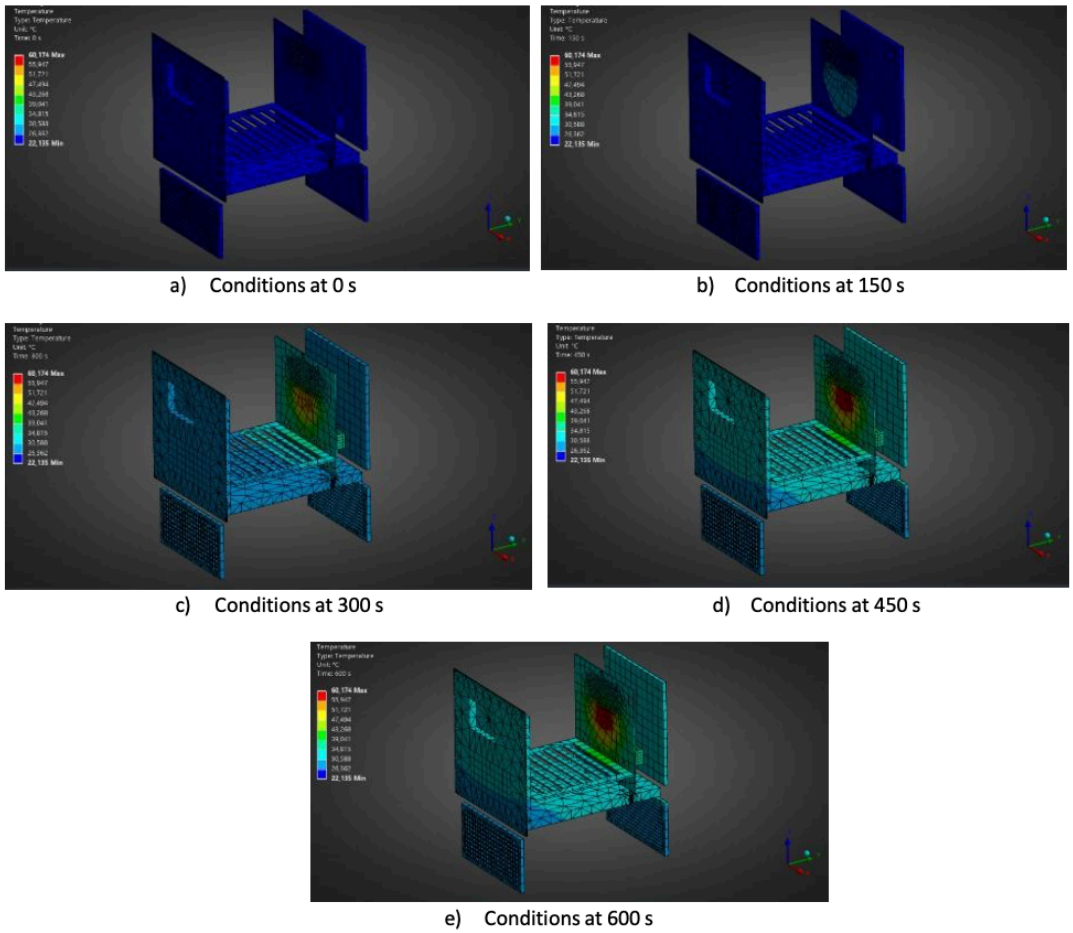



**Fig. 4.** Simulation results of heat temperature distribution under fanless conditions

Theoretically, heat transfer under these conditions can be explained by the law of convective heat transfer, which follows Newton's law of cooling. The rate of convective heat transfer is expressed as:

$q = hA(Ts - T\infty)$ (3)

where$q$ is the heat transfer rate, $h$ is the convective heat transfer coefficient, $A$ is the surface area,$Ts$ is the surface temperature of the heat source, and$T\infty$ is the ambient temperature (22°C in the simulation). Under fanless conditions, the value of $h$ is relatively small due to the absence of forced flow, so heat transfer occurs more slowly. This results in the heat distribution process becoming less effective and requiring more time to reach a near-steady-state condition.

In addition, the characteristics of fluid flow in a closed space without a fan are strongly influenced by dimensionless numbers such as the Grashof number and the Rayleigh number. These numbers represent the ratio between buoyancy forces and viscous forces in the fluid, which determines the intensity of natural convection. In general, the Grashof number can be expressed as:

$Gr = g\beta(Ts-T\infty)L3$
$v2$

(4)

where$g$ is the acceleration due to gravity, $\beta$ is the thermal expansion coefficient, $L$ is the characteristic length, and$v$ is the fluid's kinematic viscosity. In this system, the resulting Grashof number is likely to fall within a moderate range, meaning that the flow is still limited and not yet strong enough to create uniform air mixing. This reinforces the occurrence of temperature stratification within the chamber.

Based on the results of transient simulations up to 600 s, it is evident that without a fan, heat distribution tends to be non-uniform and exhibits significant temperature differences between locations. The upper part of the chamber reaches high temperatures more quickly, while the lower part lags behind in the heating process. These conditions indicate that natural convection alone is not sufficiently effective for achieving uniform temperature distribution within the thermal chamber. The implication of this finding is that a fanless design is suboptimal for battery testing applications that require temperature stability and uniformity. Therefore, additional mechanisms such as forced convection are required to improve heat distribution performance within the system.

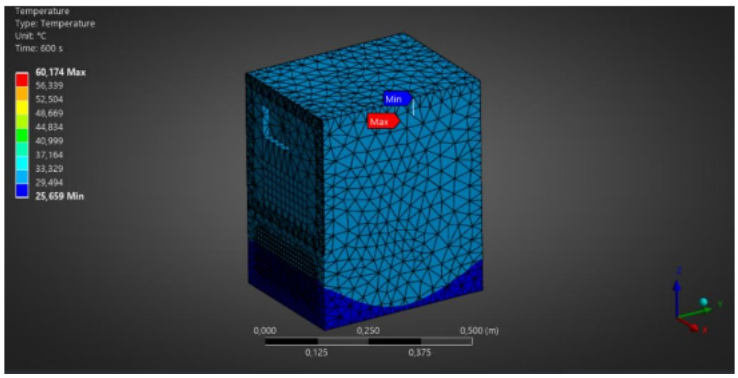



**Fig. 5.** Body temperature conditions

The figure 5 shows that the body temperature conditions during heat dissipation without a fan reveal a significant unevenness in temperature distribution throughout the thermal chamber's volume. Without forced convection, heat from the PTC heater tends to accumulate near the source and at the top of the chamber due to natural convection mechanisms. This results in a distinct temperature gradient between the top, middle, and bottom of the chamber, causing the body

temperature to be non-homogeneous. By conduction, the heat distribution within the fluid material follows Fourier's equation:

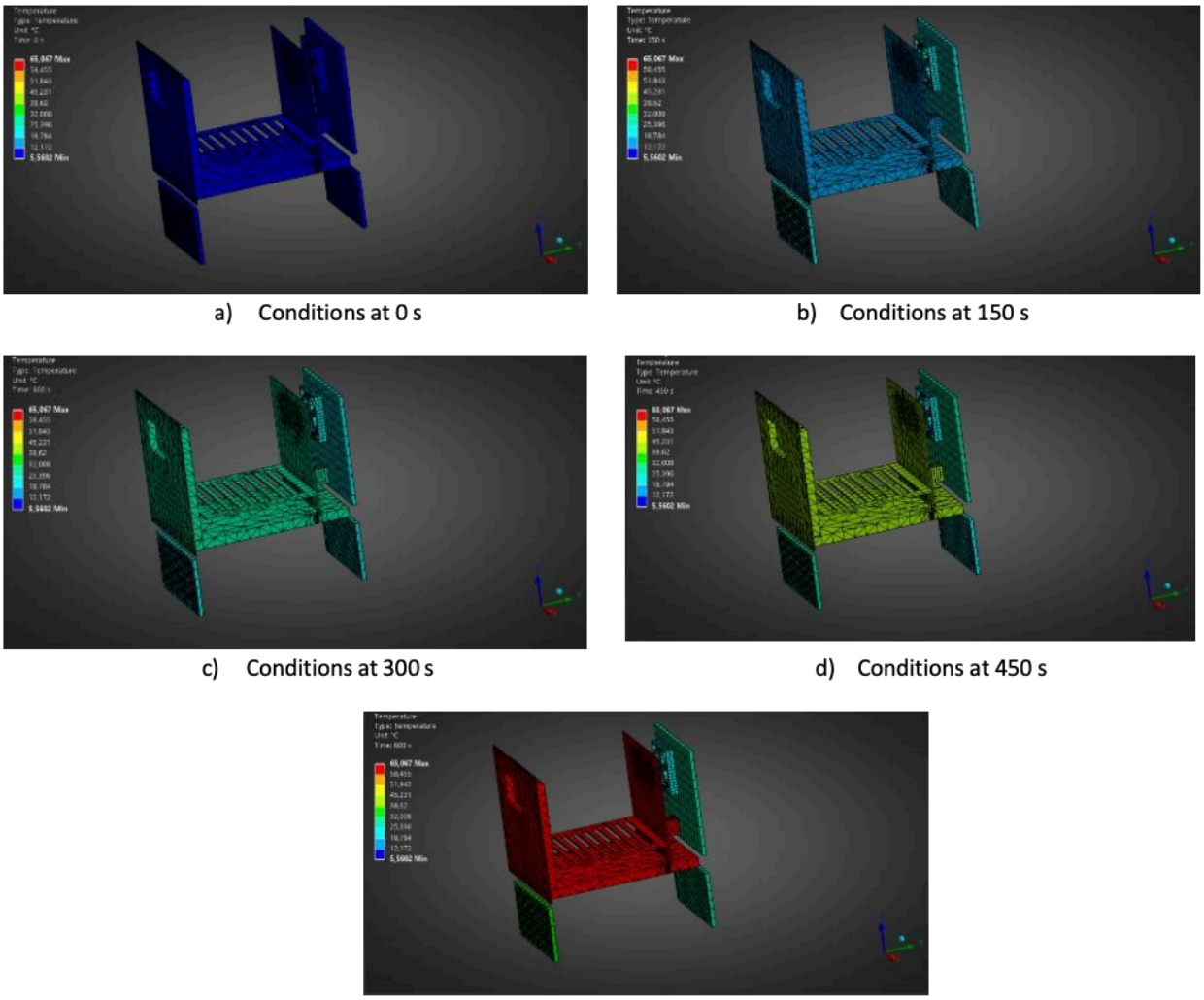
$$q = -kA \frac{dT}{dx} \quad (5)$$

Where $k$ is the thermal conductivity and $dT$ is the temperature gradient. However, because the $dx$ contribution of conduction is relatively small compared to natural convection, heat dissipation is slow and uneven. As a result, there is a potential for hotspots in certain areas that can affect the accuracy of thermal testing, especially in sensitive applications such as battery testing.

*B. Conditions with a Fan*

Under fan-assisted conditions, the temperature distribution within the thermal chamber shows significant changes compared to fanless conditions, where forced convection becomes the dominant mechanism. The airflow generated by the fan causes active fluid circulation, so that heat from the PTC heater element not only moves upward but is distributed throughout the entire chamber volume. In the initial stage (0 s to 150 s), the heat distribution begins to show a broader dispersion pattern compared to the fanless condition. As time progresses (300 s to 600 s), the temperature distribution becomes increasingly uniform, and the previously significant vertical temperature gradient begins to decrease. This indicates that the presence of the fan enhances the effectiveness of air mixing within the chamber. The simulation results are more clearly illustrated in the following figure 6.

a) Conditions at 0 s
b) Conditions at 150 s
c) Conditions at 300 s
d) Conditions at 450 s
e) Conditions at 600 s



a) Conditions at 0 s

b) Conditions at 150 s

c) Conditions at 300 s

d) Conditions at 450 s

e) Conditions at 600 s

**Fig. 6**. Simulation results of heat temperature distribution under fan conditions

Theoretically, heat transfer under these conditions still follows Newton's law of cooling, but with a higher convective heat transfer coefficient due to the presence of forced flow. The heat transfer rate can be expressed as:

$q = hA(Ts - T\infty)$ (6)

where an increase in the value of the convection coefficient ($h$) directly increases the heat transfer rate. With the fan, the airflow accelerates the distribution of thermal energy from the source throughout the space, so the time required to reach a near-steady-state condition becomes shorter. Additionally, this increase in the convection coefficient also contributes to a reduction in the temperature difference between points within the chamber.

The flow characteristics under these conditions can also be analyzed using dimensionless numbers such as the Reynolds number, which describes the flow properties resulting from the influence of the fan. The Reynolds number can be expressed as:

$$Re = \frac{\rho V L}{\mu} \quad (7)$$

Where $\rho$ is the fluid density, $V$ is the air flow velocity, $L$ is the characteristic length, and $\mu$ is the dynamic viscosity. With the fan, the Reynolds number increases, causing the flow to tend toward turbulent conditions or at least enhanced mixing, which greatly aids in temperature uniformity. This differs from conditions without a fan, which rely solely on natural convection with weaker flow intensity.

Based on simulation results up to 600 seconds, it is evident that the use of a fan produces a significantly more homogeneous temperature distribution compared to the scenario without a fan. The temperature gradient between different parts of the chamber is reduced, thereby minimizing the risk of hotspots forming. Additionally, heat dissipation occurs more rapidly and evenly throughout the entire chamber volume, indicating an improvement in overall system performance. This condition is crucial in battery testing applications, where temperature uniformity is a key parameter for ensuring the validity of test results. Therefore, the use of fans as a forced convection mechanism has proven effective in improving heat distribution quality within the thermal chamber.

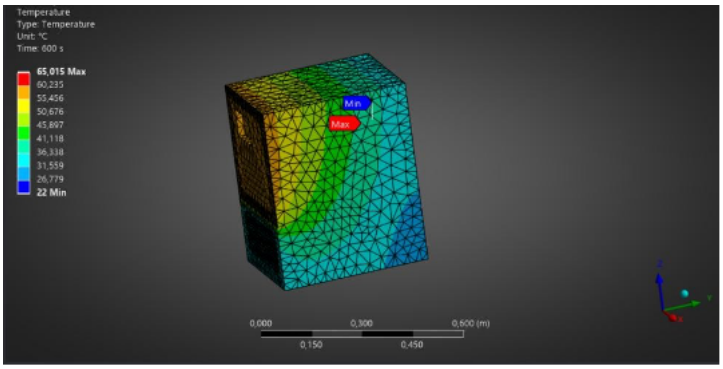



**Fig. 7**. Body temperature conditions

The figure 7 shows that the body temperature conditions during heat dissipation with a fan exhibit a much higher degree of temperature uniformity compared to conditions without a fan. The presence of forced airflow causes heat from the PTC heater to be distributed more evenly throughout the chamber volume, thereby reducing temperature differences between points. The forced convection mechanism enhances air mixing, which results in a reduction of both vertical and horizontal temperature gradients within the chamber. In terms of conduction, heat distribution still follows Fourier's law:

$$q = -kA \frac{dT}{dx} \quad (8)$$

however, the primary contribution comes from forced convection, which accelerates the transfer of thermal energy. Consequently, the body temperature tends to be homogeneous and approaches ideal conditions for thermal testing, with a significantly lower potential for hotspots to form. These conditions demonstrate that the use of a fan is effective in improving the quality of temperature distribution within the thermal chamber.

### *3.2 Cooling Distribution Analysis*

#### *A. Conditions Without Fans*

In the fanless condition, the distribution of cold temperatures within the thermal chamber indicates the dominance of natural convection, which tends to be less effective at distributing temperature evenly. In the initial stage (0 s), the cooling effect of the Peltier module begins to influence the air around the cold surface through conduction. Over time (150 s to 300 s), the air that has cooled becomes denser (density increases) and thus moves to the lower part of the chamber, while the warmer air remains at the top. This phenomenon causes the formation of a vertical temperature gradient opposite to that in the heating case, where the lower part of the chamber becomes cooler than the upper part. Consequently, the cooling distribution becomes uneven and has the potential to create localized cold spots in certain areas. The simulation results can be seen more clearly in the following figure 8.

a) Conditions at 0 s b) Conditions at 150 s
c) Conditions at 300 s d) Conditions at 450 s
e) Conditions at 600 s

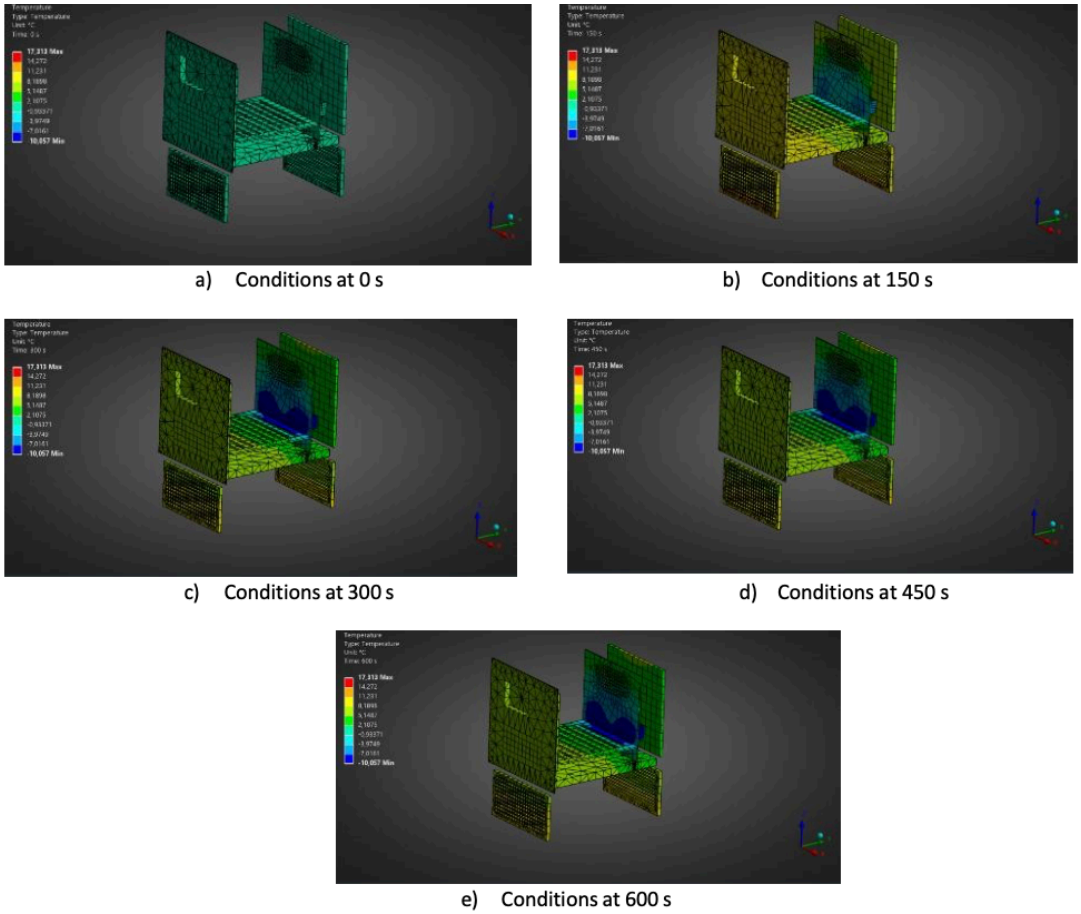



**Fig. 8.** Simulation results of cooling distribution under fanless conditions

Theoretically, heat transfer in this cooling process still follows Newton's law of cooling, with heat flowing from the fluid toward the cold surface. The heat transfer rate can be expressed as:

$q = hA(T\infty - Ts)$ (9)

Where $Ts$ is the temperature of the cold surface and $T\infty$ is the temperature of the surrounding fluid. Under fanless conditions, the value of the convective heat transfer coefficient $h$ is relatively small due to the absence of forced flow, so the process of heat release from the air to the cooling surface proceeds slowly. This results in a longer time required to reach a stable temperature distribution and increases the likelihood of temperature imbalance within the space.

In addition, the characteristics of fluid flow under these conditions are influenced by dimensionless numbers such as the Grashof number, which describes the strength of buoyancy forces resulting from density differences. The Grashof number can be expressed as:

$Gr = g\beta(T\infty - Ts)L3$
$v2$

(10)

which indicates that the greater the temperature difference, the stronger the potential for natural convection. However, in this system, the resulting flow tends to be limited and not strong enough to produce uniform air mixing. This causes cooler air to accumulate at the bottom, while the upper portion remains at a higher temperature.

Based on simulation results up to 600 s, it is evident that the cooling distribution without a fan tends to be non-uniform and exhibits significant temperature differences between locations. Cooling is more effective around the area near the cooling module and the bottom of the chamber, while the upper part experiences slower cooling. These conditions indicate that natural convection mechanisms are unable to optimally distribute the cooling effect throughout the entire chamber volume. In the context of battery testing, this can lead to inconsistent test conditions due to local temperature variations. Therefore, additional mechanisms such as forced convection are needed to improve the uniformity of cooling distribution within the thermal chamber.

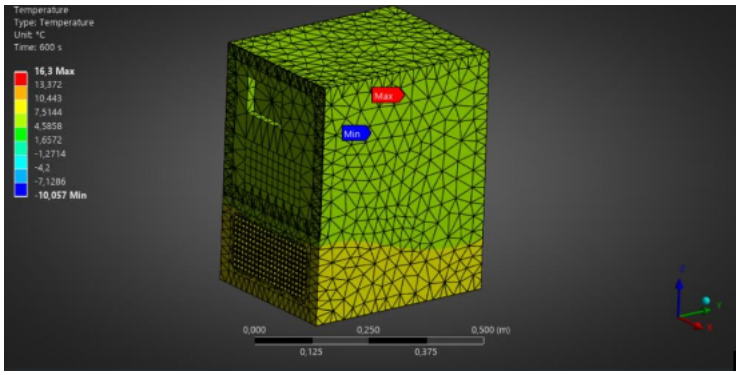



**Fig. 9.** Body temperature conditions

The figure 9 shows that the Body temperature conditions in fanless cooling systems show an uneven temperature distribution with a tendency for cold air to accumulate at the bottom of the chamber. The cooling generated by the Peltier module causes the surrounding air to decrease in temperature and increase in density, causing it to move downward due to the mechanism of natural convection. Consequently, the lower part of the chamber has a lower temperature compared to the upper part, which remains relatively warm due to the lack of air circulation. By conduction, heat transfer in a fluid follows Fourier's law:

$$q = -kA \frac{dT}{dx}$$

(11)

However, this contribution is relatively small compared to limited natural convection. This results in non-uniform body temperatures and creates cold spots in certain areas, which can affect the accuracy of thermal testing in applications such as battery testing.

*B. Conditions with a Fan*

Under fan-assisted conditions, the distribution of cool temperatures within the thermal chamber showed a significant improvement in performance compared to fanless conditions. The airflow generated by the fan creates a forced convection mechanism that accelerates heat transfer from the air to the cold surface of the Peltier module. In the initial stage (0 s to 150 s), cooling is not only localized around the module but begins to spread throughout the entire chamber volume via air circulation. As time progresses (300 s to 600 s), the temperature distribution becomes increasingly uniform, and the temperature differences between points begin to decrease. This indicates that the fan plays a crucial role in eliminating the accumulation of cold air in a single area and distributing it more widely. The simulation results can be seen more clearly in the following figure 10.

a) Conditions at 0 s
b) Conditions at 150 s
c) Conditions at 300 s
d) Conditions at 450 s
e) Conditions at 600 s

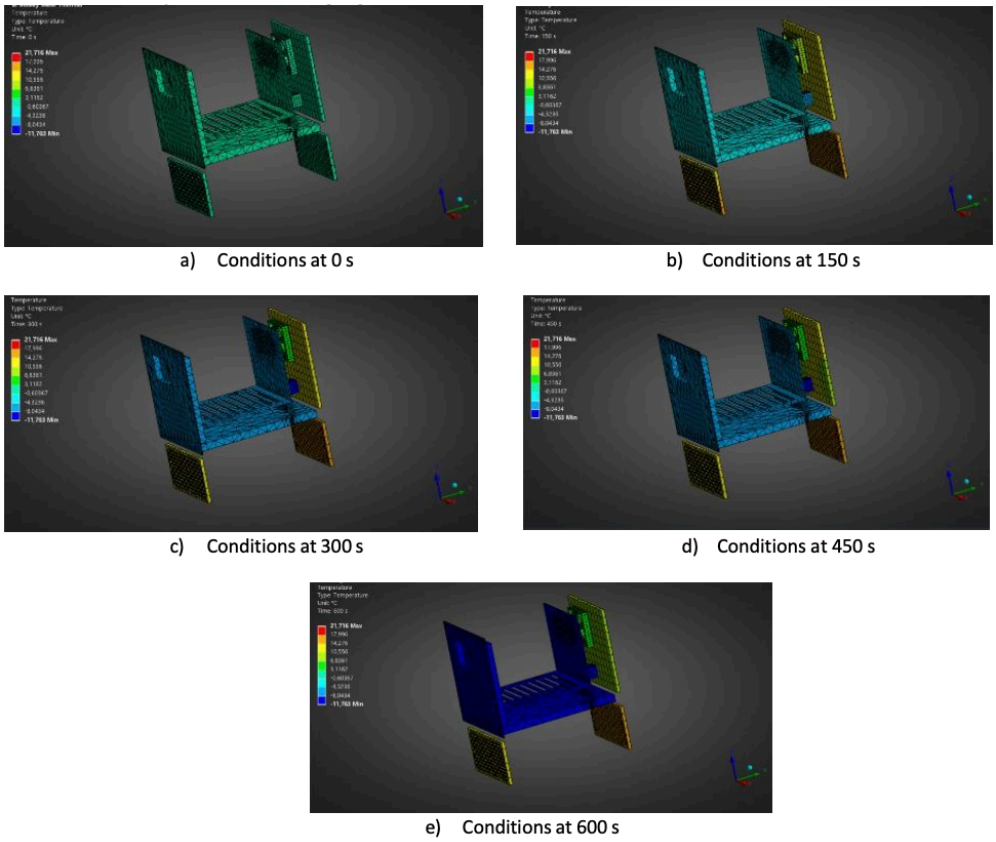



**Fig. 10.** Simulation results of cooling distribution under fan conditions

Theoretically, heat transfer under these conditions still follows Newton's law of cooling, with the heat transfer rate increasing due to the rise in the convection coefficient. This relationship can be expressed as:

$$q = hA(T\infty - Ts) \quad (12)$$

where the increase in the value of the convection coefficient ($h$) is caused by the forced airflow from the fan. As the convection coefficient increases, the process of heat transfer from the air to the cold surface becomes more effective, allowing the air temperature inside the chamber to be lowered more quickly and evenly. Additionally, continuous air circulation helps reduce the thermal boundary layer around the cooling surface.

The flow characteristics under these conditions can also be analyzed using the Reynolds number, which indicates the intensity of the flow caused by the fan. The Reynolds number equation is expressed as:

$$Re = \frac{\rho V L}{\mu} \quad (13)$$

where the increase in airflow velocity ($V$ ) caused by the fan leads to an increase in the Reynolds number. This promotes the formation of a more turbulent flow or, at the very least, enhances the mixing process within the chamber, resulting in a more homogeneous temperature distribution. Unlike conditions without a fan, fluid flow no longer depends solely on density differences but is actively controlled by the air circulation system.

Based on simulation results up to 600 seconds, it is evident that the use of fans significantly reduces the temperature gradient and results in a more uniform cooling distribution. There is no longer excessive accumulation of cold air at the bottom, as air is continuously circulated throughout the chamber. These conditions demonstrate that forced convection is effective in overcoming the limitations of natural convection in the cooling process. In the context of battery testing, a homogeneous temperature distribution is crucial for ensuring consistent test conditions. Therefore, the use of fans is an effective solution for improving the quality of cooling distribution within the thermal chamber.

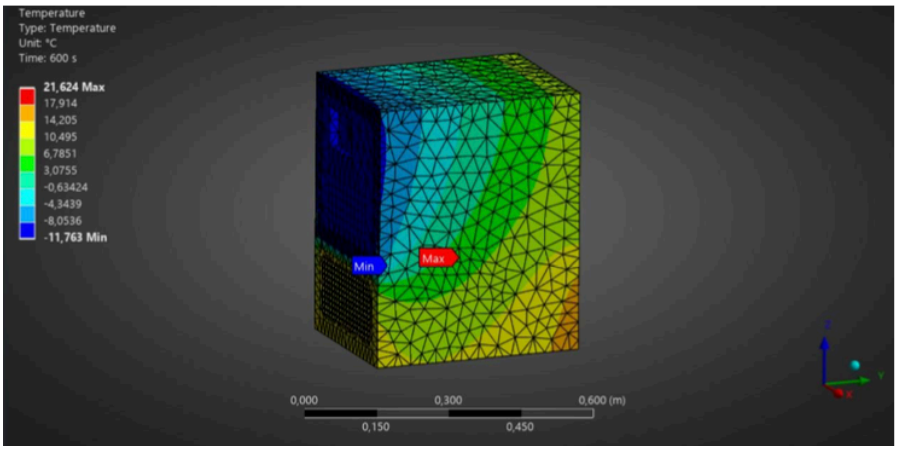



**Fig. 11.** Body temperature conditions

The figure 11 shows that the Body temperature conditions during cooling with a fan show a much more homogeneous temperature distribution compared to conditions without a fan. The forced airflow from the fan prevents the cold air generated by the Peltier module from accumulating at the bottom; instead, it is distributed evenly throughout the chamber's volume. The forced convection mechanism enhances air mixing, thereby minimizing both vertical and horizontal temperature gradients. By conduction, heat transfer still follows Fourier's law:

$$q = -kA \frac{dT}{dx}$$

(14)

However, the primary contribution comes from forced convection, which accelerates the distribution of thermal energy. As a result, the chamber temperature tends to remain stable and uniform, thereby reducing the potential for local cold spots to form. These conditions are highly conducive to thermal testing requirements that demand high temperature uniformity within the chamber.

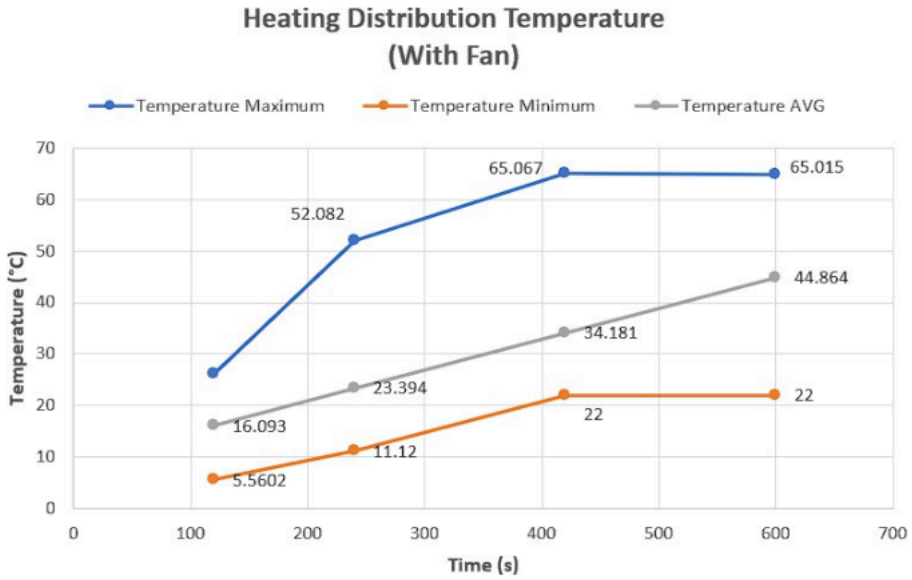
### *3.3 Comparison of Natural Convection and Forced Convection Performance*

Based on the simulation results, there is a significant difference in performance between the natural convection mechanism (without a fan) and the forced convection mechanism (with a fan) during both the heating and cooling processes inside the thermal chamber. Under fanless heating conditions using a PTC heater, the maximum temperature reached approximately 60.17°C at 360 s and tended to remain stable until 600 s, while the minimum temperature was only around 25.6°C. This indicates a considerable temperature difference (±35°C), suggesting uneven heat distribution. Conversely, under fan-assisted conditions, although the maximum temperature increased to approximately 65.01°C, the minimum temperature also increased significantly to 22°C, resulting in more uniform heat distribution throughout the chamber.

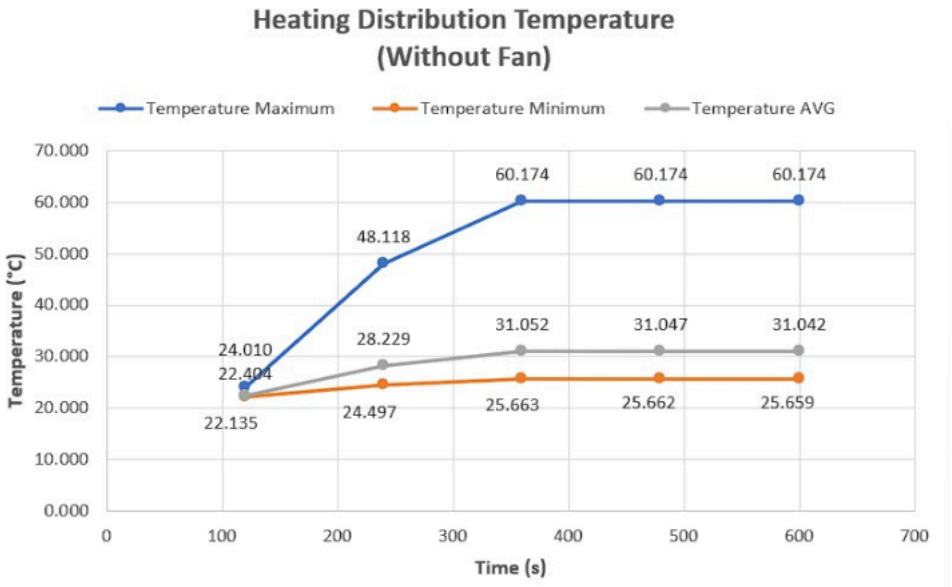
In the cooling process using a Peltier module, a similar phenomenon is observed. Under fanless conditions, the minimum temperature reaches approximately -10.05°C, but the maximum temperature remains around 16.3°C, resulting in a significant temperature difference (±26°C). This indicates that cooling is effective only in specific (local) areas, while other parts remain relatively warm. Conversely, under fan-assisted conditions, although the minimum temperature is slightly lower (-11.76°C), the maximum temperature is also better controlled at around 21.62°C, and most importantly, the average temperature drop is significantly greater, reaching 1.40°C at 600 s. This indicates that cooling is distributed more effectively throughout the entire chamber volume.

In terms of average temperature, forced convection demonstrated better performance in accelerating the system's thermal response. During heating without a fan, the average temperature reached only about 31.04°C at 600 s, whereas with a fan it rose to 44.86°C, indicating more effective heat distribution throughout the chamber. Conversely, during cooling, the average temperature without a fan remains stagnant at around 6.25°C, while with a fan it drops drastically to 1.40°C. This confirms that a fan not only accelerates distribution but also enhances the overall efficiency of energy transfer. For a clearer comparison of performance without a fan (natural convection) and with a fan (forced convection) at minimum, maximum, and average temperatures, see the following figure 12.

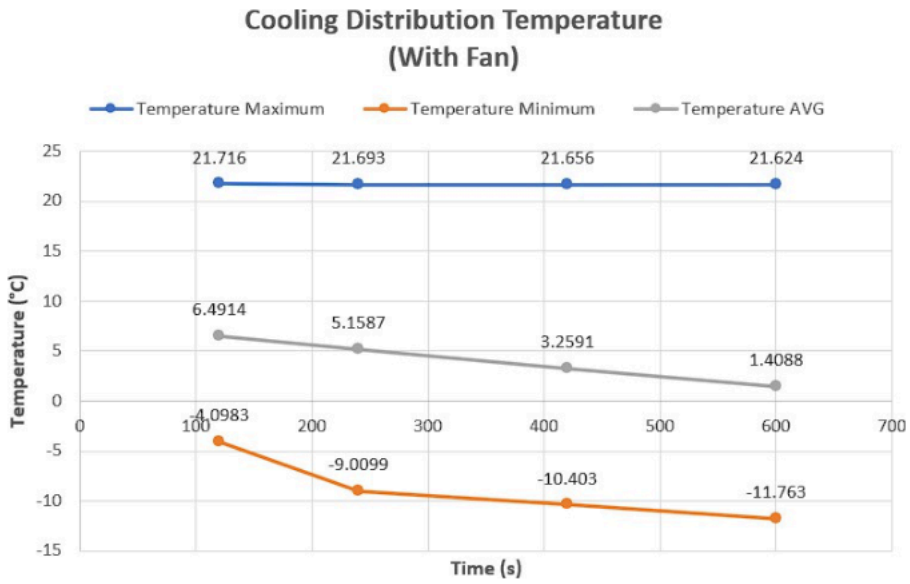
a) Heating Distrinution Temperature (With Fan)
b) Heating Distrinution Temperature (Without Fan)
c) Cooling Distrinution Temperature (With Fan)
d) Cooling Distrinution Temperature (Without Fan)



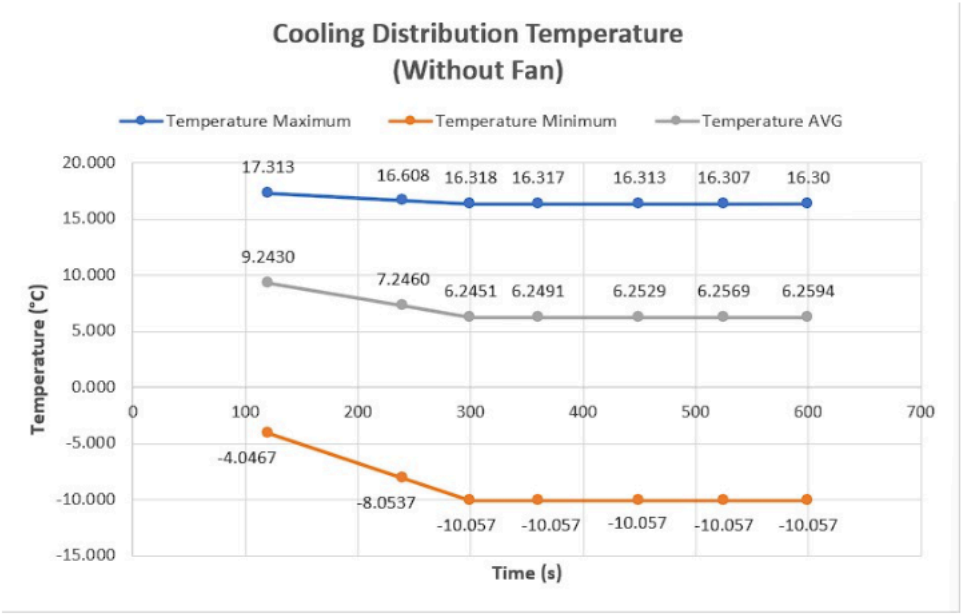

a) Heating Distrinution Temperature (With Fan)




b) Heating Distrinution Temperature (Without Fan)




c) Cooling Distrinution Temperature (With Fan)




d) Cooling Distrinution Temperature (Without Fan)

**Fig. 12.** Comparison chart of natural convection and forced convection performance

Theoretically, this difference is caused by an increase in the convective heat transfer coefficient in forced convection, which follows the relationship:

$$q = hA(Ts - T\infty)$$

where the value of the heat transfer coefficient ($h$) increases significantly due to the presence of forced airflow. In natural convection, the value of the heat transfer coefficient ($h$) is relatively small, so heat transfer is limited and depends on buoyancy. Conversely, in forced convection, the airflow generated by the fan enhances fluid mixing and accelerates temperature distribution. Based on all simulation results, it can be concluded that forced convection is far superior in producing a more

uniform temperature distribution, a faster thermal response, and higher heating and cooling efficiency. This makes the use of fans a crucial component in thermal chamber design for battery testing applications.

### *3.4 Physical Interpretation and Process Implications*

The simulation results show that the temperature distribution within the thermal chamber is significantly influenced by the heat transfer mechanisms at play, namely natural convection and forced convection. In fanless conditions, the distribution of hot and cold air is governed by buoyancy forces resulting from differences in fluid density, which leads to the formation of temperature stratification. Hot air from the PTC heater tends to rise to the top, while cold air from the Peltier module sinks to the bottom. This pattern results in an uneven vertical temperature distribution, creating zones with extreme temperatures (hotspots and cold spots) within the chamber. Physically, this indicates that natural convection has limitations in achieving effective fluid mixing.

Conversely, in systems equipped with fans, forced airflow can significantly alter temperature distribution patterns. Fans generate air circulation that enhances interaction between fluid layers, thereby reducing temperature gradients and resulting in a more homogeneous distribution. From a physics perspective, increased flow velocity leads to greater convective heat transfer intensity, which directly improves the efficiency of thermal energy distribution within the system. This not only accelerates heating and cooling processes but also reduces reliance on the stratification phenomena inherent in natural convection. Consequently, the system becomes more responsive and stable in achieving the desired temperature conditions.

The implications of these findings for the battery testing process are significant, particularly regarding the validity and consistency of test data. Uneven temperature distribution under fanless conditions can cause each test point to experience different thermal conditions, rendering the test results unrepresentative. Conversely, the use of a fan allows for the creation of a more uniform test environment, ensuring that each battery sample is subjected to relatively consistent temperature conditions. This is crucial because battery performance and characteristics are highly sensitive to temperature changes, even on a small scale. Therefore, temperature homogeneity is a key parameter in ensuring the quality of test results.

Furthermore, from a system design perspective, these results imply that fans should be used not merely as supplementary components, but as key elements in enhancing the thermal chamber's performance. Fanless designs may be simpler, but they have limitations in temperature distribution that can affect the overall effectiveness of the system. Considering the application's requirement for high temperature uniformity, the implementation of a forced convection mechanism is a more optimal solution. Therefore, integrating fans into the thermal chamber design is recommended to ensure uniform temperature distribution, rapid thermal response, and improved testing process reliability.

### *3.5 Limitations of the Study*

This study has several limitations that must be considered when interpreting the results and planning future research. One of the main limitations is the use of ANSYS-based numerical simulations, which rely heavily on assumptions and model simplifications. Parameters such as material properties, boundary conditions, and the distribution of heat and cold sources are assumed to be ideal, so they do not fully represent real-world conditions in the field. Additionally, the effects

of external factors such as heat leakage, thermal radiation, and insulation imperfections are not modeled in detail in this simulation.

Another limitation lies in the modeling of thermal components, where the PTC heater and Peltier module are simply represented as heat and cold sources with constant characteristics. In real-world conditions, the performance of these two components is significantly influenced by factors such as voltage variations, thermal efficiency, and the surrounding environmental conditions. This can lead to discrepancies between simulation results and the system's actual performance. Additionally, the effect of component performance degradation over time is not accounted for in this study.

From a fluid flow perspective, the simulation has not yet fully captured the complexity of potential turbulence phenomena, particularly under conditions involving a fan. The flow model used likely still relies on a specific approach (e.g., laminar flow or a simplified turbulence model), and thus may not be capable of representing flow interactions in detail throughout the entire chamber volume. Furthermore, variations in airflow velocity and direction resulting from the fan design have not been analyzed in depth. This has the potential to affect the accuracy of temperature distribution predictions, particularly on a local scale.

Finally, this study has not yet been supplemented with experimental validation to compare the simulation results with real-world data. Without direct testing on a thermal chamber prototype, the accuracy of the model used cannot be fully confirmed. Therefore, further research is highly recommended to conduct physical experiments to verify the simulation results and evaluate the system's performance under actual operational conditions. Consequently, the results obtained will have a higher level of reliability and can be applied more widely.

## 4. Conclusion

Based on the results of numerical simulations using ANSYS, it was found that heat transfer mechanisms significantly influence the temperature distribution within the thermal chamber. Under fanless conditions (natural convection), both the heating process using a PTC heater and the cooling process using a Peltier module exhibit uneven temperature distribution. In fanless heating, the maximum temperature reaches approximately 60.17°C, while the minimum temperature is only around 25.66°C, resulting in a significant temperature difference. Meanwhile, in fanless cooling, the minimum temperature reached -10.05°C, but the maximum temperature remained around 16.3°C, indicating cold accumulation in certain areas. This condition confirms that the natural convection mechanism has limitations in creating a homogeneous temperature distribution due to the low intensity of fluid mixing.

In contrast, under forced convection conditions, there was a significant improvement in temperature distribution performance during both heating and cooling processes. During forced convection heating, the maximum temperature increased to approximately 65.01°C, with the minimum temperature reaching 22°C, and a higher average temperature of approximately 44.86°C, indicating a more uniform heat distribution. In fan-assisted cooling, the minimum temperature reaches -11.76°C with the average temperature dropping to 1.40°C, indicating more effective and thorough cooling. Additionally, the use of fans accelerates the system's thermal response and reduces temperature gradients within the chamber. Overall, forced convection has proven superior to natural convection in producing a uniform, stable temperature distribution suitable for battery testing requirements. Therefore, the integration of fans is a critical component in thermal chamber design to enhance the quality and reliability of the testing process.

## Acknowledgement
This research was funded by a grant from Universiti Putra Malaysia - Geran Putra Inisiatif (GPI/2024/9795100).

## References

[1] Banerjee, Abhik, Xuefeng Wang, Chengcheng Fang, Erik A. Wu, and Ying Shirley Meng. "Interfaces and interphases in all-solid-state batteries with inorganic solid electrolytes." *Chemical reviews* 120, no. 14 (2020): 6878-6933. doi: https://doi.org/10.1021/acs.chemrev.0c00101

[2] Rahmat, Muhammad Aqil Afham, Adnan Ibrahim, Muhammad Amir Aziat Ishak, Ubaidah Syafiq, and Khaled M. Al-Aribe. "Numerical investigation of thermal and airflow profiles in diverse solar dryer chamber configurations." *Case Studies in Thermal Engineering* 73 (2025): 106612. https://doi.org/10.1016/j.csite.2025.106612

[3] Biçer, Emre, Ahmet Aksöz, Recep Bakar, Çağla Odabaşı, Willar Vonk, Maria Inês Soares, Rafaela Gonçalves et al. "Solid-state batteries: chemistry, battery, and thermal management system, battery assembly, and applications—a critical review." *Batteries* 11, no. 6 (2025): 212. https://doi.org/10.3390/batteries11060212

[4] Chaffer, Ryan T., Joseph Johnson, Matthew J. DiDomizio, and Mark B. McKinnon. "Anaerobic Pyrolysis Chamber: An atmosphere-controlled furnace for controlled-atmosphere thermal decomposition." *HardwareX* (2025): e00716. https://doi.org/10.1016/j.ohx.2025.e00716

[5] Bilal, S., Imtiaz Ali Shah, A. S. Alqahtani, and M. Y. Malik. "Significance of aspect ratio of internal heating element in optimizing thermal production of water with ternary nano composition in crown chamber." *Case Studies in Thermal Engineering* 68 (2025): 105882. doi: https://doi.org/10.1016/j.csite.2025.105882

[6] Fu, Songqiao, and Hongmao Qin. "Optimization of thermoelectric module configuration and cooling performance in thermoelectric-based battery thermal management system." *World Electric Vehicle Journal* 16, no. 7 (2025): 344. https://doi.org/10.3390/wevj16070344

[7] Lyu, You, Abu Raihan Mohammad Siddique, S. Andrew Gadsden, and Shohel Mahmud. "Experimental investigation of thermoelectric cooling for a new battery pack design in a copper holder." *Results in Engineering* 10 (2021): 100214. https://doi.org/10.1016/j.rineng.2021.100214

[8] García, Antonio, Carlos Micó, Mariany Chávez Cobo, Imad Elkourchi, and Jagadish Babu Vemula. "A comprehensive methodology for characterization and electro-thermal modelling for a next-generation solid-state battery." *Energy* 322 (2025): 135601. https://doi.org/10.1016/j.energy.2025.135601

[9] Pfaff, Jonas, Matilda Fransson, Ludovic Broche, Mark Buckwell, Donal P. Finegan, Stefan Moser, Sebastian Schopferer, Siegfried Nau, Paul R. Shearing, and Alexander Rack. "In situ chamber for studying battery failure using high-speed synchrotron radiography." *Synchrotron Radiation* 30, no. 1 (2023): 192-199. doi: https://doi.org/10.1107/S1600577522010244

[10] Song, Lulu, Zhenhua Wang, Yongchun Zhu, Zhizuan Zhou, Boxuan Wang, Dong Wang, Ulises Rojas Alva, Lizhong Yang, and Xiaoyu Ju. "A review of safety issues in lithium-ion battery transportation process: Research advances and challenges." *Journal of Loss Prevention in the Process Industries* (2026): 105912. doi: https://doi.org/10.1016/j.jlp.2026.105912

[11] Mariello, Massimo, Elisa Scarpa, Luciana Algieri, Francesco Guido, Vincenzo Mariano Mastronardi, Antonio Qualtieri, and Massimo De Vittorio. "Novel flexible triboelectric nanogenerator based on metallized porous PDMS and parylene C." *Energies* 13, no. 7 (2020): 1625. doi: https://doi.org/10.3390/en13071625

[12] Olabi, A. G., Hussein M. Maghrabie, Ohood Hameed Kadhim Adhari, Enas Taha Sayed, Bashria AA Yousef, Tareq Salameh, Mohammed Kamil, and Mohammad Ali Abdelkareem. "Battery thermal management systems: Recent progress and challenges." *International Journal of Thermofluids* 15 (2022): 100171. doi: https://doi.org/10.1016/j.ijft.2022.100171

[13] Buntarto, Electric Motorcycles, p. 160, 2016.

[14] S. A. Hussien, A. B. M. Ali, O. J. Alkhatib, and I. Mahariq, "Enhanced passive thermal management of lithium -ion batteries with a conical cylindrical chamber incorporating various phase change materials," Sci. Rep., vol. 15, no. 1,pp. 1-16, 2025. doi: https://doi.org/10.1038/s41598-025-19597-0

[15] Sun, Shu, Theodore S. Rappaport, Timothy A. Thomas, Amitava Ghosh, Huan C. Nguyen, Istvan Z. Kovacs, Ignacio Rodriguez, Ozge Koymen, and Andrzej Partyka. "Investigation of prediction accuracy, sensitivity, and parameter stability of large-scale propagation path loss models for 5G wireless communications." *IEEE transactions on vehicular technology* 65, no. 5 (2016): 2843-2860. doi: https://doi.org/10.1109/TVT.2016.2543139

[16] Luthfi, Muhammad, Muhamad Ghozali, and Yudhy Kurniawan. "The effect of battery settings and fan parameters on the performance of the lithium-ion battery thermal management system in a circular configuration." *JST (Jurnal Sains dan Teknologi)* 13, no. 3 (2024): 362-373. https://doi.org/10.23887/jstundiksha.v13i3.84849
[17] Y. Zhao, "Zhao2021PhD.pdf," November 2020.
[18] Akgül, Mehmet Bahattin. "Evaluation of the thermal performance of vapor chambers integrated skived fin heat sink for PV-inverter cooling under multi-heat source condition." *Case Studies in Thermal Engineering* (2025): 106729. https://doi.org/10.1016/j.csite.2025.106729
[19] Dai, Fang, and Mei Cai. "Best practices in lithium battery cell preparation and evaluation." *Communications Materials* 3, no. 1 (2022): 64. https://doi.org/10.1038/s43246-022-00286-8
[20] Cortês, Andreia, Diana Gonçalves, António Tadeu, and João Almeida. "Investigating the thermal and acoustic performance of a modular living wall: a case study using bioclimatic and reverberation chambers." *Case Studies in Thermal Engineering* (2025): 107372. doi: https://doi.org/10.1016/j.csite.2025.107372
[21] Jiayue, C. H. E. N., and L. I. Guofu. "RESEARCH ON THE STAGGERED BATTERY THERMAL MANAGEMENT SYSTEM BASED ON AIR DISTRIBUTION PIPES." *Thermal Science* 29, no. 3B (2025): 2421.doi: https://doi.org/10.2298/TSCI240803249C
[22] Yao, Ping, and Xuewen Liu. "Electrochemical and thermal analysis of lithium-ion batteries based on variable solid-state diffusion coefficient concept." *World Electric Vehicle Journal* 15, no. 9 (2024): 416. https://doi.org/10.3390/wevj15090416
[23] Mahek, Mehwish Khan, Mohammad Alkhedher, Mohammed Ghazal, Mohammad Ali Abdelkareem, Mohamad Ramadan, and Abdul-Ghani Olabi. "Effects of control volume outlet variation on axial air cooling of lithium-ion batteries." *International Journal of Thermofluids* 19 (2023): 100373.
[24] Tao, Zhenxiang, Haozhi Zhou, Yuyao Cao, Xubo Huang, Wei Wang, and Hui Zhang. "A review of thermal runaway gases and detection methods for lithium-ion batteries." *Thermal Science and Engineering Progress* (2025): 104427.doi: https://doi.org/10.1016/j.tsep.2025.104427
[25] P. I. M. Keyser et al., "Battery Thermal Modeling and Testing Energy Storage Task Lead: Ahmad Pesaran NREL Energy Storage Program Computer-Aided Engineering of Batteries (CAEBAT)," Energy, no. May, 2011.
[26] Luo, Yu, Aibing Zhang, Dandan Pang, and Baolin Wang. "Dynamic performance analysis of a thermoelectric cooling-based thermal management system for lithium-ion power batteries." *Case Studies in Thermal Engineering* 72 (2025): 106315. https://doi.org/10.1016/j.csite.2025.106315
[27] Del Nostro, Pierluigi, Gerhard Goldbeck, Ferry Kienberger, Manuel Moertelmaier, Andrea Pozzi, Nawfal Al-Zubaidi-R-Smith, and Daniele Toti. "Battery testing ontology: An EMMO-based semantic framework for representing knowledge in battery testing and battery quality control." *Computers in Industry* 164 (2025): 104203. https://doi.org/10.1016/j.compind.2024.104203
[28] ABA, Muhammad Ulin Nuha, Bayu Wahyudi, and Mohamad Sofie. "Perancangan Pemantauan Suhu Climatic Chamber Berbasis Arduino Dilengkapi Heater Dan Peltier." *Elkom: Jurnal Elektronika dan Komputer* 14, no. 1 (2021): 105-113.
[29] Saqli, Khadija, Houda Bouchareb, Mohammed Oudghiri, and Kouider Nacer M'SIRDI. "Lithium-ion battery thermal parameter identification and core temperature estimation." *Politeknik Dergisi* (2023): 1-1. https://doi.org/10.2339/politeknik.1161986
[30] Touki, Hasan Douha, Md Nobin Hossain, Ahmed Imtiaz Ferdous, Yaminul Hoque, Mohiuddin Ahmed, and Sumon Saha. "Thermal management of micropolar ferrofluid forced convective flow in a T-shaped vented chamber via a proportional-integral-derivative controller." *South African Journal of Chemical Engineering* (2025).doi: https://doi.org/10.1016/j.sajce.2025.11.011
[31] Hildebrand, Stephan, Asma Eddarir, and Natalia Lebedeva. "Overview of battery safety tests in standards for stationary battery energy storage systems." *Publications Office of the European Union: Luxembourg* (2024). doi: https://doi.org/10.2760/81430
[32] Kristiyono, Roedy, Budi Nugroho, and Bambang Supriyanto. "Automatic Charging Battery Lithium Untuk Kendaraan Listrik." *Jurnal Teknika* 7, no. 4 (2022): 236-242. https://doi.org/10.52561/teknika.v7i4.195
[33] Pandeeswari, Jayaraman, Gunamony Jenisha, Kumlachew Zelalem Walle, and Masashi Kotobuki. "Recent research progress on all-solid-state Mg batteries." *Batteries* 9, no. 12 (2023): 570.doi: https://doi.org/10.3390/batteries9120570
[34] Adit, Md Ahnaf, Samiul Hasan, and Nirendra Nath Mustafi. "Performance analysis of a novel battery thermal management system integrating thermoelectric and dielectric immersion cooling in EVs." *Energy Conversion and Management: X* (2026): 101550. https://doi.org/10.1016/j.ecmx.2026.101550
[35] A. A. Yaqien, M. Yamin, and C. P. Mahandari, "LiFePO4 Battery Thermal Management System Using Mini-Channel Cooling Plates for Electric Vehicle Applications," JST (Journal of Science and Technology), vol. 12, no. 3, pp. 779-789, 2024. https://doi.org/10.23887/jstundiksha.v12i3.59241

[36] Al Qubeissi, Mansour, Ayob Mahmoud, Moustafa Al-Damook, Ali Almshahy, Zinedine Khatir, Hakan Serhad Soyhan, and Raja Mazuir Raja Ahsan Shah. "Comparative analysis of battery thermal management system using biodiesel fuels." *Energies* 16, no. 1 (2023): 565. doi: https://doi.org/10.3390/en16010565

[37] Gómez Díaz, Kenia Yadira, Susana Estefany De León Aldaco, Jesus Aguayo Alquicira, Mario Ponce Silva, Samuel Portillo Contreras, and Oscar Sánchez Vargas. "Thermal management systems for lithium-ion batteries for electric vehicles: A review." *World Electric Vehicle Journal* 16, no. 7 (2025): 346. https://doi.org/10.3390/wevj16070346

[38] Sabri, M., Nguyen Thi Bich Ngoc, Utkarsh Kumar, Ade Irwan, Fadly Ahmad Kurniawan Nasution, Muhammad Amir Yusuf Harahap, Muhammad Gilang Husni Lubis et al. "PVDF composites reinforced by dispersed Al2O3 particles for thermally and mechanically stable separators in lithium-ion battery." *Next Materials* 11 (2026): 101676. https://doi.org/10.1016/j.nxmate.2026.101676

[39] Niu, Zhe-hui, Kai-ge Pang, Bin-bin Pang, and Shuai Wang. "Orthogonal experimental-based thermal management design and simulation optimization of a liquid-cooled battery module." *Scientific Reports* 15, no. 1 (2025): 33442. https://doi.org/10.1038/s41598-025-19098-0

[40] Mahek, Mehwish Khan, Mohamad Ramadan, Sharul Sham bin Dol, Mohammed Ghazal, and Mohammad Alkhedher. "A comprehensive review of thermoelectric cooling technologies for enhanced thermal management in lithium-ion battery systems." *Heliyon* 10, no. 24 (2024). https://doi.org/10.1016/j.heliyon.2024.e40649