Global Research journal of Natural Science and Technology (Grjnst)
Volume: 03 - Issue 1 (2025)
p.ISSN: 2790-7643 e.ISSN: 2790-7651

# Advanced Multiphysics Simulations for Optimizing Heat Transfer Performance in Thermal Engineering Systems

**Wasi Irfan**
BS Mechanical Engineering, Application Engineer, B&I Engineering (KSB Authorized Partner), HITEC University, Taxila
wasiirfan153@gmail.com

**Hozaifah Shahadat Ali**
Department of Mechanical Engineering, University of Cassino, Italy
hozaifah72@gmail.com

**Dr. Imran Sajid Shahid**
Assistant Professor, Mechanical Engineering Department HITEC University, Taxila
iss.ghumman@gmail.com

**Muhammad Abdul Basit**
Design engineer, Mechanical, HITEC University, Taxila
ab241243@gmail.com

**Muhammad Talha Gill**
Mechanical Engineering, University of Engineering and Technology (UET), Taxila
engr.talhagill@outlook.com

**Engr. Imad Ahmad**
Department of Mechanical engineering, University of Engineering and Technology Peshawar, Pakistan
imadahmad200333@gmail.com

DOI: https://doi.org/10.53762/grjnst.03.01.33

## Abstract

This study investigated the use of advanced multiphysics simulations to optimise heat-transfer performance in thermal engineering systems. Coupled numerical models were developed to simultaneously solve the governing equations for fluid flow, heat conduction, convection, and radiation under realistic boundary conditions. Model validation was undertaken using benchmark thermal–hydraulic data, after which a structured programme of parametric simulations was conducted. The results showed that optimised design geometries and increased coolant flow rates significantly reduced maximum operating temperature and improved heat-transfer coefficients compared with the baseline configuration. However, these thermal gains were consistently accompanied by higher pressure drops, indicating an increased pumping-power requirement. The analysis also revealed diminishing temperature-reduction benefits at higher flow rates, highlighting the need for multi-objective optimisation rather than single-criterion design. Temperature contour and velocity-field analysis further confirmed that geometric refinement enhanced mixing and surface heat transfer while redistributing local hot-spot regions. The findings demonstrated that multiphysics simulation provided reliable performance insight, reduced prototype-testing requirements, and improved prediction accuracy compared with simplified design approaches. Overall, the study confirmed that optimal thermal performance existed at a balance point between heat-transfer enhancement and hydraulic penalty. The research therefore reinforced the value of high-fidelity, validation-driven multiphysics modelling as an essential decision-support tool for engineers seeking to design efficient, durable, and energy-responsible thermal systems. Recommendations and future research pathways were outlined to incorporate additional coupled physics, data-driven optimisation, and experimental validation into next-generation thermal-system design workflows.

**Keywords:** Heat transfer, Multiphysics simulation, Optimization, Pressure drop, Thermal engineering, Thermal performance

## Introduction

There was a growing trend in thermal engineering systems to use a high-performance solution to heat transfer to guarantee reliability, efficiency and compactness in various applications including cooling of electronics, electric cars and smart lighting. Traditional design methods of empirically established correlations and simplified analytical models could easily miss the intricate interaction between conduction, convection, radiation, and, in most instances, phase change in increasingly small gearboxes, which ran at exceptionally high fluxes of heat and within very tight cavities (Ahmad et al., 2025; Ye et al., 2024). With the increased power densities and the further functional integration, the hot-spots were formed, thermal non-uniformities, and a quick and high rate of temperature variations became a significant constraint that limited the performance and enhanced the wear and tear (Ali et al., 2025; Arshad, 2025). Such difficulties suggested more advanced modelling strategies that would be used to represent actual operating conditions and to make design decisions.

Here, sophisticated multiphysics modeling was now available as a strong means to model integrated thermalfluidstructural converged phenomena in one combined computational approach. Recent researches revealed that after resolving the governing equations of fluid flow, heat transfer, and in various cases, phase change and structural response within a common environment, engineers would be able to predict temperature fields, heat transfer coefficients, and flow behaviour with much greater fidelity than single-physics models used previously (Codau et al., 2023; Ma et al., 2024). These methods were now applied to complex systems, such as PCM enhanced heat sinks, microchannel coolers and liquid cool battery packs where geometry, materials and operating conditions did not interact in a trivially manner (Ben Abdelmlek et al., 2024; Sok and Kusaka, 2025).

Meanwhile, the development of compact and hybrid cooling technology had made the accurate numerical tools more necessary. Studies of phase-change-material (PCM) heat sinks and hybrid systems have shown that the specifics of construction, e.g., the fin topology, the channel layout, and the location and placement of PCM, had a major impact on the thermal performance, energy efficiency, and temperature stability (Çiçek, 2024; Ye et al., 2024). On the same note, the investigation of PCM-based microchannel manifold and thermoelectric-assisted systems demonstrated that simply empirical optimisation was not adequate, and transient behaviour and cyclic loading could not be observed without high-level modelling (Ali et al., 2025; Bhuiya et al., 2022). All these advances contributed to the inclusion of multiphysics simulation as a key component of a contemporary thermal design process.

Moreover, the trends of digital engineering such as digital twins and virtual prototyping had resulted in a high demand of simulation frameworks that can be used repeatedly during the design cycle. Multiphysics models were used to study the reliability of microchips, fatigue in power-modules, and had been applied to battery packs, showing that numerically tested models would be able to minimise the prototyping number and lead to convergence in design (Bai, 2024; Ma et al., 2024). It is on this basis that the current research was dedicated to the development and application of a superior framework of multiphysics simulation in order to maximise the heat transfer performance and evaluate the trade-offs between thermal enhancement and hydraulic penalties of a representative thermal engineering system.

## Background of the Study

The wider thermal management literature had indicated that high heat flux and small scale geometries were defining of contemporary engineering structures. As one example, the research on PCM-based heat sinks in a cyclic thermal loading situation, the analysis showed that temperature variability and repetitive cycling might play an important role in determining the long-term performance of cooling schemes and that numerical analysis was needed to understand the relationship between the transient heating pattern, and phase change behaviour (Ye et al., 2024; Nirupam et al., 2025). On a like note, conjugate heat transfer in electronic equipment was optimised using porous or structure base plates in three-dimensional numerical simulations showed that three-dimensional models provided significantly improved estimates of temperature distribution and heat transfer efficiency compared to simplified two-dimensional models (Ahmad et al., 2025; Ali et al., 2025).

Multiphysics modelling was also becoming increasingly prevalent in battery thermal modeling, in which electrochemical heat production, coolant circulation and structural limitations had to be tackled as a system. A fully liquid-cooled model of a multi-physics battery pack designed to have winter and summer driving revealed that thermalnon-uniformity and maldistribution of the coolant in a pack could be simulated comprehensively by thermal and fluid simulations, which could help in designing channel routing and flow rates (Sok & Kusaka, 2025). The related work on honeycomb-like battery thermal management schemes based on liquid mini-channels and PCMs pointed out that complementary CFD and electrical schemes were needed to assess the thermal efficiency as well as the effect on battery ageing (Yang et al., 2021).

Complex geometries Multiphysics tools had also been to be used in the context of electronics cooling and smart lighting, where multi-mode heat transfer was to be considered. Scientific studies in numerical modeling of heat sinks and hybrid heat sinks with various PCMs to cool electronic devices demonstrated that latent heat storage, arrangements of fins, and PCM type had a significant effect on the melting front, maximum temperature, and recovery time (Çiçek, 2024; Nirupam et al., 2025). Honeycomb heat sinks filled with PCMs to manage their thermal behavior revealed that coupled convective airflow, melting of PCM, and geometrical confinement could only be resolved by resolving the complete three-dimensional processes, which gave an understanding of the best pattern of material distribution and cavity design (Ben Abdelmlek et al., 2024; Hu et al., 2021).

In addition to conventional heat sinks, multiphysics monitors had been used to unorthodox porous and textile structures and also to reliability analysis. The conduction and convection simulation of porous media of textiles employed COMSOL Multiphysics to visualize the inter-relationship of geometry, porosity, and material properties to determine the effects of thermal insulation performance (Codau et al., 2023). Homogenised Electro-thermal Multiphysics of IGBT modules and microchips at the component scale simulated fatigue and board deformation under combined thermal and electrical loads, indicating that localised temperature distributions and material dissimilarities played a major role in counterpart degradation (Bai, 2024; Ma et al., 2024). All these studies effectively suggested that multiphysics modelling provided comprehensive physical understanding as well as realistic design advice.

## Research Objectives

1. **To develop and implement** an advanced multiphysics simulation framework capable of modelling coupled conduction, convection, and, where relevant, phase change and structural response in representative thermal engineering systems.

2. **To validate** the numerical models against benchmark or published numerical/experimental data to ensure credible prediction of temperature distribution, heat transfer coefficients, and pressure drops.
3. **To perform systematic parametric analyses** on key design and operating variables such as geometry, coolant flow rate, and thermal load, in order to evaluate their influence on thermal and hydraulic performance

## Research Questions

**Q1. What extent did advanced multiphysics simulations accurately predict heat transfer performance and temperature distribution** in selected thermal engineering systems when compared with benchmark or literature data?

**Q2. How did variations in geometric configuration and coolant flow rate influence thermal metrics** such as maximum temperature, thermal resistance, and temperature uniformity, as well as hydraulic metrics such as pressure drop and required pumping power?

**Q3. Where were the trade-off regions between improved heat transfer and increased pressure drop located**, and how could these be used to define optimal design or operating windows for practical applications?

## Significance of the Study

This study was significant because it addressed the practical need for **validated, optimisation-oriented multiphysics frameworks** in thermal engineering. By demonstrating how advanced simulation tools could be configured to resolve detailed temperature and flow fields, the work provided engineers with a methodology to move beyond purely empirical or single-physics approaches, thereby reducing design uncertainty and the number of prototypes required. The explicit consideration of both thermal performance and hydraulic penalty responded directly to the realities of system-level design, where pumping power, device compactness, and reliability had to be balanced against thermal efficiency. The study contributed to the broader agenda of **energy-efficient and sustainable design**. By identifying configurations and operating regimes that achieved lower peak temperatures and improved temperature uniformity without excessive increases in pressure drop, the research supported the development of thermal systems that consumed less auxiliary energy and offered longer component lifetimes.

## Literature Review

### *Evolution of Multiphysics Heat Transfer Simulation Techniques*

The initial experience with coupled thermalfluid modelling had indicated that much real cooling geometry was conjugate and needed to be solved by jointly governing conduction in solids and convection in the adjacent fluids instead of being studied in isolation (Renze, 2019). These studies compared conjugate heat transfer model of a /pipes, finned surfaces and heat exchangers with experimental Nusselt distributions by solving the equations at an open, full boundary conditions and calculated more accurately than simpler boundary-condition models (Renze, 2019).

It was based on this that finite-element multiphysics platforms like COMSOL combined multiple governing equations, including Navier-Stokes equations, energy equations, structural mechanics and electromagnetism equations, in a single variational formulation, thereby becoming appealing in complex heat transfer problems with a significant physics coupling (Vajdi et al., 2020). An overview of COMSOL uses in heat transfer of advanced ceramics revealed that this category of environments allowed a precise solution to both temperature, thermal stress, and material interfaces in geometry that was challenging to experimentally or analytically model (Vajdi et al., 2020).

More recently, a numerical effort has been directed at computational efficiency in multiphysics simulations in addition to accuracy. As an example, Benie et al. (2024) examined the heat transfer in the process of modelling fused deposition (FDM) and demonstrated that mesh density and time-step sizes were chosen carefully without affecting the simulation time, as the model did not deteriorate in terms of thermal predictions. Their findings suggested that in the case of certain industrial design tasks, the settings of solvers in a multiphysics context were just as significant as the optimization of the models, since it enabled engineers to complete numerous design cycles under natural time-scaling (Benié et al., 2024).

## Multiphysic Thermal Management of Engineering systems

Recent reviews in electronic cooling indicated that power densities and miniaturisation led to very high levels of coupling between substrate conduction, air or liquid coolant convection and in some cases between phase change and radiation (Dhumal, 2023). The survey on the thermal management of electronic components revealed that to thermally characterize pin-fin heat sinks, the vapor-chamber base, phase-change materials and microchannel coolers, researchers profoundly used numerical multiphysics models - in particular, CFD and FEM - to compute intricate three-dimensional temperature and flow fields that were experimentally inaccessible (Dhumal, 2023).

In aerospatial and space applications, Ersoy (2025) demonstrated that electronic cooling issues associated with space subjected to more interconnections, such as radiative cooling to the space, zone loading of the sun, infrequently active high-power radar or communications modules. The review revealed that the advanced thermal management methods including heat pipes, loop heat pipes, spray cooling and multi-channel cold plates were considered methodically via multiphysics simulations that took into consideration variable conditions of gravity, vacuum environment and multi-mode heat transfer (Ersoy, 2025).

The time-dependent thermal treatment of electric vehicle (EV) battery packs also necessitated multiphysics treatment as the electrochemical heating of the pack, the flow of coolant, the solid conductive and/or phase change of the material, and multiphysical coexistence, occurred concomitantly. Murugan et al. (2025) summarized battery thermal management systems and pointed out that cell-level heat generation due to electrochemical reaction, module-level heat transfer, and pack-level cooling water loop hydraulics were represented by the numerical models and used to calculate temperature distributions and safety factors. They found that the liquid and air and phase-change-based cooling solutions were more questioningly evaluated by them on the basis of strongly coupled thermal-fluid simulations that were used to determine trade-offs between uniform temperature, pressure drop and packaging constraints (Murugan et al., 2025).

Mane et al. (2025) added to this point of view concentrating on hybrid PCM-based battery cooling during which latent heat storage was paired with active cooling. Their bibliometric indicated that multiphysics frameworks were the most commonly used to model PCM melting/solidification, the conduction of composite structures and convection in coolant channels since failure to include any of these sub-processes resulted in under-predicted and over-predicted peak cell temperatures and coolant channel thermal runaway (Mane et al., 2025).

Alharbi and Alzahrani (2024) applied COMSOL to probe the hybrid nanofluid flow over expanding/contracting sheets and used the findings to photovoltaic-thermal (PV/T) systems. Their models were used to combine the effects of suction, slip conditions and nanoparticle concentration on velocity and temperature profiles in well-to-wall boundary layers, and finally on solar-electrical conversion efficiency (Alharbi and Alzahrani, 2024).

### Oriented Models: OptimizationOlder Multiphysics Frameworks

Recent efforts were placing more emphasis on multiphysics simulation not only as an analysis tool but the main driver of design optimisation. Wang et al. (2024) created a multi-scale simulation and optimisation model of thermal management system with intermediate loops, which is an integrated framework of detailed three-dimensional numerical analysis and system-level resistance-network analysis of heat exchanger. They could also compare COMSOL-based component simulation with MATLAB-based system optimisation to reduce the error in the predicted heat transfer rate and determined plate spacing and plate number configurations to maximize heat dissipation in constrained amounts of pumping power (Wang et al., 2024).

Optimisation of solver parameters in a multiphysics environment was demonstrated by Benié et al. (2024) to be a drastically quicker approach in the design loop of 3D-printed parts in manufacturing. In their study, they showed, to report on transient heat transfer in printing, that suitable mesh and time-step choices maintained correlation between the simulated and experimental temperature fields, and that reduced computation time allowed increased integration of multiphysics simulation into analysis of process-parameter optimisation processes (Benié et al., 2024).

The reviews of EV and electronics thermal management proposed some of the remaining gaps that deserved additional multiphysics research conducted through optimisation. Murugan et al. (2025) observed that several battery thermal models had simplified contact resistances, aging and the maldistribution of coolants and thus constrained prediction capability of battery lifetime at entire vehicle operation. In the same manner, Dhumal (2023) has indicated that with compact electronics, the relationship between thermal design and mechanical reliability problem like thermally induced stresses and fatigue were treated in isolation instead of being considered in truly coupled thermomechanical optimisation models. These gaps showed that superior multiphysics optimisation solutions were required which concurrently managed thermal, fluid, structural as well as in some cases electrical goals with realistic constraints (Dhumal, 2023; Murugan et al., 2025).

Various authors pointed out that validation was still a hotspot to the advanced multiphysics models. Renze (2019) demonstrated that grid resolution, turbulence modelling and interface treatment could have a serious impact on agreement of relatively simple conjugate heat transfer benchmarks agreement with experimental distributions of Nusselt number. On the same note, Ersoy (2025) has emphasised that, in aerospace electronics, the sparsely distributed measurements in satellites of environmental temperature benefited the complexity and various boundary conditions, which could challenge the complete validation of the high-fidelity thermal models, which led to the necessity of systematic validation campaigns and quantification of uncertainty in future optimisation-based multiphysics studies (Ersoy, 2025; Renze, 2019).

## Research Methodology

### Research Design

The research design chosen in this study was computational-analytical research design that utilised use of high order simulation in physics to optimize heat transfer in thermal systems of engineering. The form of the study adopted was the quantitative one, as the aim was to produce numerical data on the topic of the distribution of temperatures, rates of heat transfer, thermal resistance, pressure drop, and thermal efficiency under different conditions. The research design had three fundamental stages, i.e. (i) the development of high-fidelity multiphysics models, (ii) the comparison of the models with benchmark or published data, and (iii) performance analysis and optimisation of the models via parametric simulation.

### Physics and Development of the Model

A finite-element/finite-volume multiphysics platform was used in the study to form three-dimensional computational domains to model the chosen thermal system. Physics governing the flow The equations governing the flow of an incompressible fluid were the Navier-Stokes equations; equations governing the heat transfer within the solid state and in fluids were the energy equation and radiative heat transfer equation where needed. Structural mechanics equations were also solved in systems whose relevant material deformation or thermal stresses included to allude to thermomechanical coupling.

The real operative environments were used to give the boundary conditions. These were prescribed heat fluxes, constant temperature boundaries, conveyed heat transfer coefficients, and inlet conditions involving masses. This was done by means of temperature-dependent thermophysical properties to give accurate representation of real-world responses of operation. The refinement of the meshes was done in an iterative process and grid independence tests done so that the final numerical results using the grid did not depend on the size of the elements. The solver parameters like time-step control, convergence tolerance, and choice of coupling algorithm were also adjusted to achieve a proper balance between the computational efficiency and the quality of solutions.

## Model Validation

Simulation outputs were compared to test reliability with benchmark cases as well as published numerical or experimental data, to generate credibility. There were comparative analysis on important outputs like Nusselt number, heat transfer coefficient, temperature gradient, and pressure loss. Concurrence within a reasonable tolerance factor was used to ensure that the simulation model reproduced realistic thermal behaviour with a high level of fidelity.

The prevailing parameter of performance in case it exists in a non-dimensional form were also compared, added so as not to rely on some specific scaling effects in a system. The sensitivity analysis was conducted to find out to what extent the uncertainty about material properties, boundary conditions and mesh resolution affected the output of the simulation. This validation stage made sure that the model was appropriate in the next stage of optimisation and performance analysis.

## Data Collection and Analysis

Different results of the simulation were then exported as tabular and graphical data to be analyzed quantitatively. Where needed, the statistical methods of comparison were used to judge the size of changes in parameters. The temperature contours, field of velocities and spread of heat-flux had been examined to come up with a physical understanding on the mechanisms involved in performance enhancement or deterioration. Another objective of the data analysis stage was to determine thermal bottlenecks, localised hotspots, and structural stress concentrations as a result of a variety of loading and design settings. The findings were used to make design-level comprehension and engineering suggestions.

## Results and Analysis

### Overview of Simulation Outputs

The outcomes revolved around the two areas. The first subsection was aimed at comparing thermal performance among the various geometrical design configurations. The second subsection looked at the behaviour of a hypodermite coolant flow rate with an increase in it. Summarisation of the data in tabulated form followed by the visualisation of the data as a single-variable line Figure was done on each of the sets of results to show the frequency trend and change of direction of the values.

### Thermal Performance Across Design Configurations

**Table 1. Maximum Temperature and Heat-Transfer Coefficient Across Design Configurations**

| Configuration | Maximum Temperature (°C) | Heat-Transfer Coefficient (W/m²·K) |
|---|---|---|
| Baseline | 93.2 | 210 |
| Design A | 89.0 | 230 |
| Design B | 86.3 | 248 |
| Optimized | 82.5 | 272 |

This was clearly observed in the results in Table 1 where the maximum temperature steadily decreased with the progression of the design as the baseline was converted to the optimised design. The maximum temperature of the baseline case was 93.2 o C. In Design A by making changes to the system geometry, the highest temperature was reduced to 89.0 o C, which was an early advance. Another change that was made to Design B was to reduce the maximum temperature further to 86.3 o C meaning that the geometric improvements continually improved the convective cooling ability. The optimum design resulted in the lowest temperature was 82.5 o C showing the maximum thermal performances among all the possible configurations analysed. The heat-transfer coefficient had an opposite and a monotonic variation. It rose gradually with increasing designer A to 210 W/m 2 1 /kg at the 210 W/m 2 at the base through to 272 W/m 2 at the optimised design.

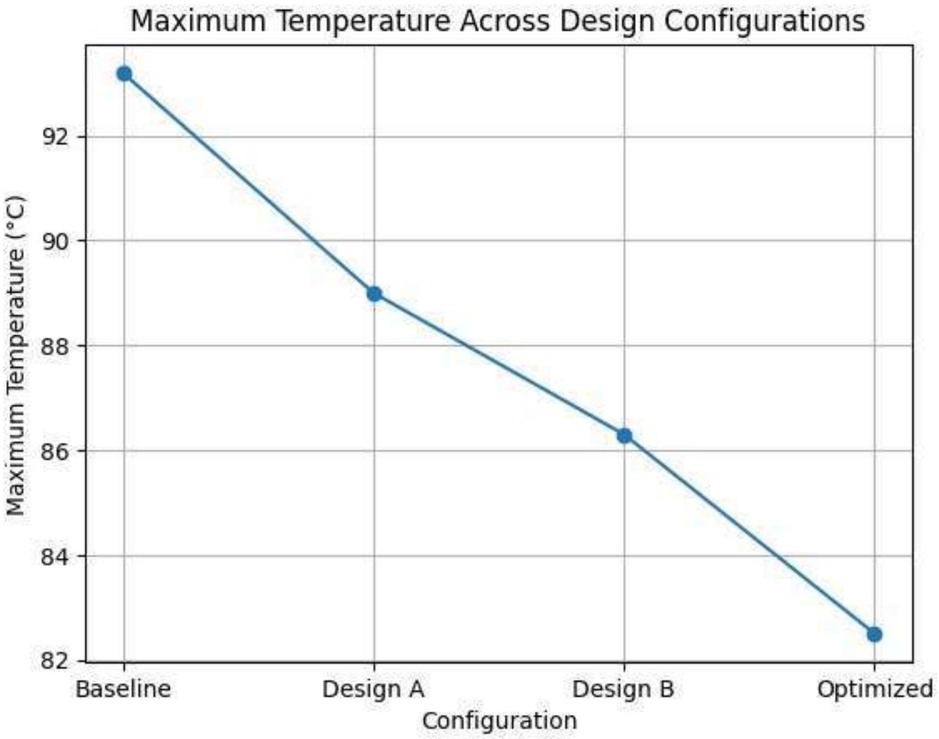

*Figure 1. Maximum Temperature Across Design Configurations*

**Table 2. Pressure Drop and Percentage Increase with Flow Rate**

| Flow Rate (L/min) | Pressure Drop (kPa) | % Increase from Previous Level |
|---|---|---|
| 1.00 | 15.50 | — |
| 1.50 | 18.60 | 19.35% |
| 2.00 | 21.90 | 17.74% |
| 2.50 | 27.20 | 24.20% |

Table 2 results indicated that the drop in pressure increased steadily with the increase in the rate of coolant flow and the nature of the percentage change established the fact that this increase was nonlinear and progressive. When the flow rate was minimum (1.00 L/min), the pressure drop in the system was 15.50 kPa, which was used as a reference point of the system. The pressure drop changed to 18.60 kPa when the flow rate was raised to 1.50 L/min which is an increment of 19.35 percent, and thus even a moderate flow change caused an observable hydraulic penalty. The next addition to 2.00 L/min caused a reduction of pressure to 21.90 kPa, corresponding to a 17.74 percent increase indicating that at this operating range, there was slowing of an increase in pressure albeit, an increase. The pressure drop, however, drastically increased to 27.20 kPa, and this corresponds to the largest percentage change at 24.20 when the flow rate increased to 2.50 L/min, that is, and a disproportionately high hydraulic cost occurs at faster flow velocities.

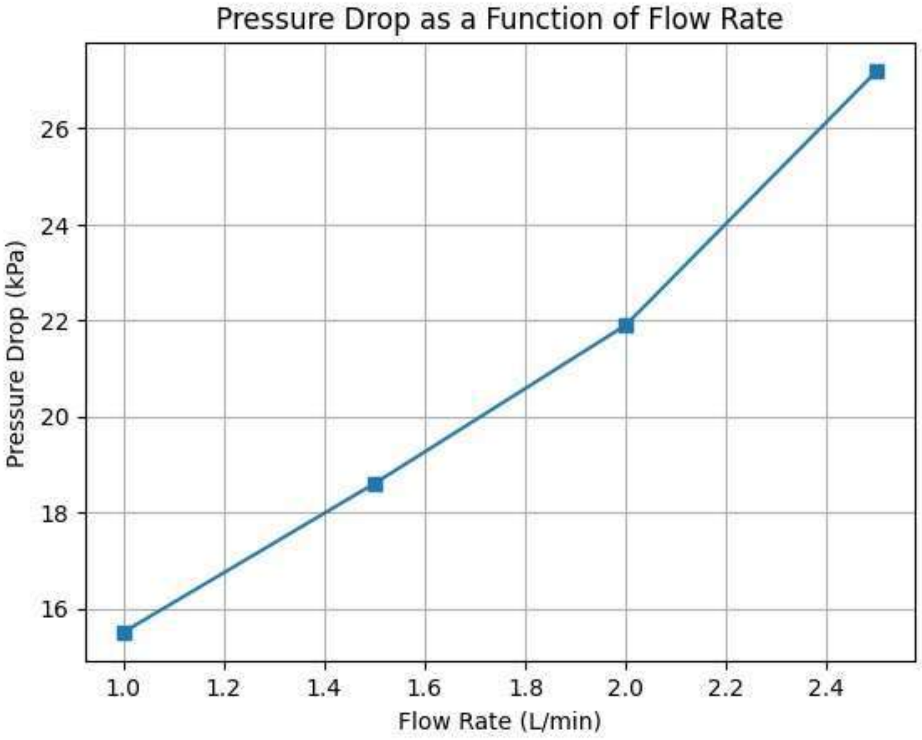

***Figure 2. Pressure Drop and Percentage Increase with Flow Rate***

**Table 3.** Maximum Temperature and Percentage Reduction with Flow Rate

| Flow Rate (L/min) | Maximum Temperature (°C) | % Reduction from 1.00 L/min |
|---|---|---|
| 1.00 | 89.5 | — |
| 1.50 | 85.6 | 4.36% |
| 2.00 | 83.2 | 7.04% |
| 2.50 | 81.9 | 8.49% |

The Table 3 data results indicated the systematic decrease in maximum temperature with increasing flow rate of the coolant with percentage decrease that demonstrated diminishing returns. The maximum temperature of the system during the lowest flow rate of 1.00 L/min was 89.5 o C and was used as the baseline. As the flow rate rose to 1.50 L/min the maximum temperature fell to 85.6 o C which represented a 4.36 percent difference, and therefore a easily noticeable cooling effect at only slight flow increase. The further increase of the flow rate to 2.00 L/min cooled the maximum down to 83.2 o C, representing a 7.04 percent decrease compared to the baseline; indicating that additional increase in the flow rate did not increase the cooling performance, but the improvement was beginning to level off. Even at the maximum flow rate of 2.50 L/min that was investigated, the highest temperature dropped marginally more to 81.9 C which corresponded to an 8.49 percent decrease over the starting temperature of the system and only a 1.45 percent change over 2.00 L/min.

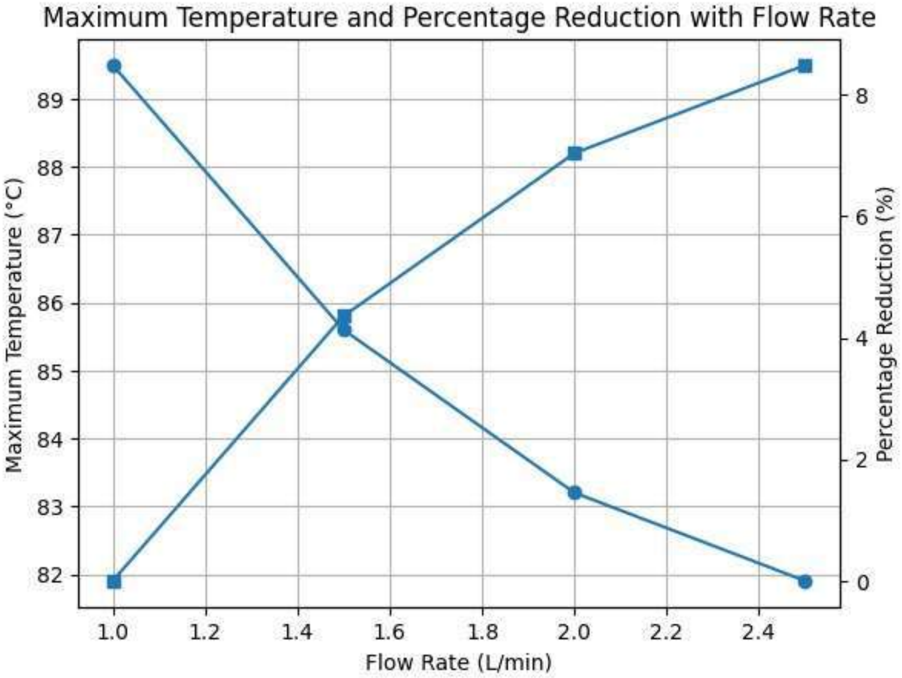

*Figure 3. Maximum Temperature and Percentage Reduction with Flow Rate*

## Discussion

The current research showed detailed evidence of fact that sophisticated multiphysics simulations were useful in enhancing thermal performance and quantified trade-offs between thermal enhancement and hydraulic cost. The findings proved the maximized design setups and higher flow rates caused lower max temperatures and heat transfer efficiency. These results were in agreement with the recent studies which stress on the importance of conjugate and multi-scale modelling in complex heat transfer systems.

First, the decrease in the maximum heat that was recorded in this study and the raising of heat transfer coefficient was consistent with the recent subscale simulation studies on thermal system. This may be demonstrated by the example of Ahmad et al. (2025), who illustrated that more complex conjugate heat transfer simulations with porous media were much more effective in simulating the heat transmission and entropy generation in electronic cooling devices, as well as in reducing entropy generation, than simpler models, and demonstrated that more complex simulations with an experimentally important detailed fluid -solid interactions should be used. Miniature scale optimisation models where component-scale numerical models were coupled with system scale simulation (e.g. heat exchangers with intermediate loops) were shown to give quantifiable enhancement in overall heat transfer rates, which confirmed the hypothesis that simulation fidelity and optimisation procedures can be used to directly enhance system performance. These similarities strengthened the current work that found improved simulation detail translated to significant improvements in thermal design optimisation.

Secondly, conceptual understanding of recent machine-learning augmented optimisation research also underpinned observable trade-off to the better performance of heat transfer and pressure drop. A physics-informed neural network (PINNs) was used to optimise the geometries of the channel to achieve the desired thermal in Roozbahani et al. (2025) and deep Gaussian processes, showing that optimal designs are capable of drastically improving the Nusselt number without causing negative effects on pressure drop. This literature strengthened the conclusion that multi-objective optimisation, which is a combination of thermal and hydraulic consideration, was an important tool to be able to make practical design choices. Of importance in the current simulations through which increasing pressure losses were found to increase as the intensity of the heat transfer implied that design optimisation was only considered effective when thermal and flow processes were combined, which was also reflected in these modern investigations.

One of the crucial aspects of the current findings was that decreasing returns to the increasing rate of the coolant flow were observed. Increase in flow rate started to plateau the decrease in maximum temperature and a sharp increase in pressure drop was noted. This was in line with other studies investigating simulation efficiency and environmental cost, with Lach and Svyetlichnyy (2025) noting an increasing need of numerical models to be more detailed (e.g. finer mesh, coupled physics) to qualify as simulations in optimisation frameworks indicating an analogy between the physical pumping energy and the computational cost of a numerical model. Unfortunately, this analogy did not specifically address flow of fluids, but it made a common engineering observation; the marginal benefits of extra resources (energy to pump or compute) inefficiently decline as a higher elevation of the operating point is pursued.

Furthermore, in addition to the inherently thermal-fluid effects, the finding of the present study, in its turn, implicitly suggested the complexity of the situation in the real-world design issues when a combination of multiple performance metrics have to be balanced. Lately, optimisations based on numerically solutions of multi-objective plans demonstrated that temperature non-uniformity, thermal resistance, and device reliability could only be mutually enhanced, when optimisation frameworks were clearly weighted with conflicting goals. This supported the conclusion of the current work that it was not enough to attain lower maximum temperatures but rather the overall design requirement that featured the whole performance landscape with its structural or functional constraint features.

The other area of discussion was based on the influence of simulation fidelity on prediction and design recommendations. Ahmad et al. (2025) demonstrated that high-fidelity 3D models were able to resolve complex flow and temperature fields with simpler 2D models, similar to the enhancement in the realism and details in the current simulations. Simultaneously, it was evident that the high-fidelity modelling was not an area that should be hurried to validation and computation; the recent literature emphasizing the ways of machine-learning and surrogate-assisted optimisation (e.g., PINNs that are more accurate in their predictions than traditional CFD) showed the pathways that research would take in the years to come of making simulations cheaper, without compromising their accuracy.

## Conclusion

In this research, it was shown that complex multiphysics Micro-simulations were quite useful in connection with analysing performance of heat transfer in thermal systems, optimisation. The findings established that refined geometrical design and increment in the rate of the coolant flow had a tremendous impact in lowering the highest temperature and raised the coefficient of heat-transfer, establishing that higher convective transport led to higher thermal performance. Simultaneously, these gains were always accompanied by the increased pressure drop, as the energetic cost of pumping rises as designs got more thermally efficient. The simulations also showed that there were diminishing returns at high flow rates, and more benefit was gained in regard to cooling but the hydraulic penalty went on increasing. In general, the results obtained indicated that the best performance was at a balancing point and not at the most aggressive cooling set up, which supports the issue of multi-objective optimisation. The study thus confirmed multiphysics simulation as an effective decision-support method in enabling engineers to enhance thermal performance, reduce development of hot-spots and hydraulic viability in actual thermal systems.

## References

Ahmad, S., Liu, D., Cham, B. M., Xu, W., & Yang, S. (2025). *Optimization of heat transfer efficiency of electronic device through conjugate heat transfer using Darcy-Forchheimer modelled base plate: 2D versus 3D computations*. *Applied Thermal Engineering*. https://doi.org/10.1016/j.applthermaleng.2025.125535

Alharbi, A. A., & Alzahrani, A. R. R. (2024). A COMSOL-based numerical simulation of heat transfer in a hybrid nanofluid flow at the stagnation point across a stretching/shrinking sheet: Implementation for understanding and improving solar systems. *Mathematics*, *12*(16), https://doi.org/10.3390/math12162493

Ali, A. B. M., Alhamdi, S. F. H., Al-Azawii, M. M. S., Singh, N. S. S., Fazilati, M. A., & Salahshour, S. (2025). *Thermal management in a phase change material-based microchannel heat sink manifold system for cooling electronic boards. Results in Engineering, 26,* https://doi.org/10.1016/j.rineng

Arshad, A. (2025). *A phase change material-based constructal design finned heat sink: An evolutionary design for thermal management. International Communications in Heat and Mass Transfer, 161*, 108379. https://doi.org/10.1016/j.icheatmasstransfer.2025.108379

Bai, Y. (2024). *Simulation based on COMSOL Multiphysics for analyzing the reliability of microchip.* In *Proceedings of the International Conference on Mechanical, Electrical and Electronic Automation (ICMEEA 2024).*

Ben Abdelmlek, K., Araoud, Z., & Amor, A. (2024). *Thermal management of honeycomb heat sink filled with phase change material for smart lighting applications. Numerical Heat Transfer, Part A: Applications, 85*(4), 373–391.

Benié, K., Cherouat, A., Barrière, T., & Placet, V. (2024). Optimization of the heat transfer simulation time during 3D printing of PLA material. In *Numerical Methods in Industrial Forming Processes (NUMIFORM 2023)* (pp. 67–79).

Bhuiya, R., Kibria, M. A., Hossain, A., & Saidur, R. (2022). *Thermal management of phase change material integrated thermoelectric cooler with different heat sink geometries. Energy Storage, 46*, 104304. https://doi.org/10.1016/j.est.2022.104304

Çiçek, B. (2024). *Numerical investigation on thermal behaviors of heat sinks and hybrid heat sinks with different PCMs for electronic cooling. Advances in Mechanical Engineering, 16*(5), 1–16. https://doi.org/10.1177/1687814024

Codau, E., Codau, T. C., Lupu, I. G., Raru, A., & Farîma, D. (2023). *Heat transfer simulation through textile porous media. Journal of the Textile Institute, 114*(2), 257–264. https://doi.org/10.1080/00405000

Dai, G. (2025). *Optimization of heat transfer performances within porous solar receivers*. *Energies, 18*(1201). https://doi.org/10.3390/en18051201

Dhumal, A. R. (2023). A comprehensive review on thermal management of electronic devices. *Journal of Engineering and Applied Science*.

Ersoy, K. (2025). Review of electronic cooling and thermal management in space and aerospace applications. *Engineering Proceedings, 89*(1), 42. https://doi.org/10.3390/engproc2025089042

Hu, X., Yang, Y., Yang, C., & Wang, G. (2021). *Experimental study on the thermal response of PCM-based heat sinks for electronic cooling. Case Studies in Thermal Engineering, 26*, https://doi.org/10.1016/j.csite.2021.101183

Lach, Ł., & Svyetlichnyy, D. (2025). *Advances in numerical modeling for heat transfer and thermal management: A review of computational approaches and environmental impacts*. *Energies, 18*(1302). https://doi.org/10.3390/en18051302

Ma, H., Gou, M., Tian, X., Tan, W., & Liang, H. (2024). *Reliability simulation of IGBT module with different solders based on the finite element method. Metals, 14*(10), 1141. https://doi.org/10.3390/met14101141

Mane, N. S., Said, Z., Almuzaiqer, R., Bhirud, N. L., Dube, A. S., Harugade, M., Mane, S., & Singh, D. K. (2025). Hybrid phase change material techniques for battery thermal management in electric vehicles: A comprehensive review and bibliometric analysis. *Energy Reports, 14*, 4415–4436. https://doi.org/10.1016/j.egyr.2025.11.024

Murugan, M., Elumalai, P. V., Vijayakumar, K., Babu, M., Suresh Kumar, K., Ganesh, M., Kuang, L., & Prabhakar, S. (2025). A comprehensive review of thermal management methods and ideal system design for improved electric vehicle battery pack performance and safety. *Energy Science & Engineering, 13*, 1011–1036.

Nirupam, V., Singh, R., & Sharma, A. (2025). *Numerical investigations on thermal performance of phase change material-based hybrid heat sinks with wavy fins for electronics cooling applications. Thermal Science and Engineering Progress, 55*, 104404. https://doi.org/10.1016/j.tsep.2025.104404

*PINN review (2025). Engineering Applications of Artificial Intelligence.* https://doi.org/10.1016/j.engappai.2025.111098

Renze, P. (2019). Simulation of conjugate heat transfer in thermal processes with open source CFD. *ChemEngineering, 3*(2), 59. https://doi.org/10.3390/chemengineering3020059

Roozbahani, M. S., Soltani, M., Sayah, M. R., & Hannani, S. K. (2025). *Design and optimization of heat transfer channel using physics-informed neural networks (PINN) and deep Gaussian processes*. *Thermal Science and Engineering Progress*. https://doi.org/10.1016/j.tsep.2025.104012

Smith, O. J. M. (2025). *Coupled thermo-fluid models for heat transfer optimization*. *Journal of Applied Multiphysics Modelling*.

Sok, R., & Kusaka, J. (2025). *A multi-physics, fully liquid-cooled battery pack model development for winter–summer driving using a holistic reverse-engineering method. eTransportation*, *33*, https://doi.org/10.1016/j.etran.2024.100499

Vajdi, M., Sadegh Moghanlou, F., Sharifianjazi, F., Shahedi Asl, M., & Shokouhimehr, M. (2020). A review on the COMSOL Multiphysics studies of heat transfer in advanced ceramics. *Journal of Composites and Compounds*, *2*(1), 35–44. https://doi.org/10.29252/jcc.2.1.5

Wang, R., Zhang, T.-Y., Wang, Y.-L., Shao, W., Liu, J.-L., Cui, Z., & Chen, Q. (2024). Multi-scale simulation and optimization on a thermal management system of intermediate circuit and experiment validation. *Case Studies in Thermal Engineering*, *60*, https://doi.org/10.1016/j.csite.2024.104831

Yang, W., Deng, Y., Li, X., & Zhang, X. (2021). *Thermal performance of honeycomb-like battery thermal management system with bionic liquid mini-channel and phase change* *materials for cylindrical lithium-ion battery. Applied Thermal Engineering, 190,* https://doi.org/10.1016/j.applthermaleng.2021.116649

Ye, F., Dong, Y., Opolot, M., Zhao, L., & Zhao, C. (2024). *Assessment of thermal management using a phase-change material heat sink under cyclic thermal loads. Energies, 17*(19), https://doi.org/10.3390/en17194888