GSJ: Volume 12, Issue 9, September 2024, Online: ISSN 2320-9186
www.globalscientificjournal.com

COOLING EFFECTIVENESS OF TUBED AND RACTANGULAR COOLING PLATES USING CFD ANALYSIS

Authors; Imran Hussain[1]

[1]Mehran university of Engineering and Technology Sindh Pakistan- 76060 Pakistan

Corresponding Authors:

1. Imran Husain (engr_imranmughal@yahoo.com)

KeyWords
Cooling plates, weiland cooling modules, cfd, ansys

ABSTRACT

Thorough experiences into the tubed cooling plate's warm conductivity and stream conduct were gotten from the CFD recreation directed under states of 320 kW/m² of intensity transition and 0.02 m/s of stream speed. The intensity source temperature is marginally higher in the Wieland CP 4009 T (300.442°C) contrasted with the miniature cooling plate (300.2 K). This distinction could show a more proficient intensity move in the Wieland plate plan. The temperature at the plate zone is additionally marginally higher for the Wieland plate (300.31°C) contrasted with the miniature cooling plate (300.13°C), which could propose better intensity dispersal or lower warm obstruction in the Wieland plate. The line temperature is higher in the miniature cooling plate (300.38°C) contrasted with the Wieland plate (300.14°C). This demonstrates that the miniature cooling plate could encounter higher warm misfortunes along the line.

The delta temperature is comparative for the two plates yet somewhat higher in the miniature cooling plate (300.2 K) contrasted with the Wieland plate (300.104°C), recommending an expected distinction in stream rate or intensity moves effectiveness at the bay. The power source temperature is altogether higher in the Wieland CP 4009 T (355°C) contrasted with the miniature cooling plate (300.2 K). This proposes that the Wieland plate is more successful at eliminating heat, potentially because of its plan or material properties

1. INTRODUCTION

The objective of this task is to push the limits of warm administration innovation by making and developing a fluid cooled heat sink framework that works on existing methodologies as well as meets the issues given by present day mechanical gadgets. The review addresses a significant advancement from customary air-cooling draws near[1]. The Fluid Cooled Intensity Sink takes another system that exploits the amazing intensity move potential. A thermally receptive cooling board is gone through a cylinder containing fluid, which delivers the fluid cooled heat safeguard. Yet again to make a cooling circle, the temperature from the component is moved from the virus surface to the liquid utilizing tubes, where it is coursed through a trade of intensity[2]. CFD examination will be done by means of the ANSYS Familiar programming. The computational outcome will be examined with the other cooling plate plans. In the wake of examining the outcomes, to build productivity of the cooling execution models might should be changed and yet again dissected with the reference information[3].

2. INTENSIVE LITERATURE REVIEW

Since the fast advancement of electronic innovation, electronic machines and gadgets currently are generally in our everyday existence. Under the state of multifunction, high clock speed, contracting bundle size, and higher power dispersals, the intensity transition per unit region expanded decisively throughout recent years[22]. In addition, the functioning temperature of the electronic parts might surpass the ideal temperature level. Hence, the successful evacuations of intensity dispersals and keeping up with the pass on at a safe working temperature play had a significant impact in protecting a solid situation of electronic parts. There are numerous techniques in hardware cooling, for example, stream impingement cooling heat pipe. Customary gadgets cooling typically involved impinging plane with heat sink showing predominance as far as unit value, weight and dependability. Thusly, the most widely recognized method for upgrading the air-cooling is through the use of impinging air jets on an intensity sink. To plan a powerful intensity sink, a few rules, for example, an enormous intensity move rate, a low tension drop, a more straightforward assembling, a less difficult construction, a sensible cost, etc ought to be thought of[23].

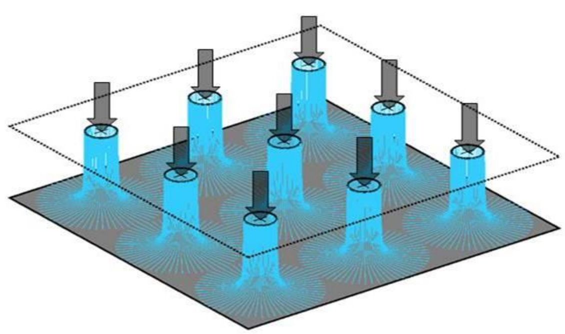


Figure 1 jet impingement cooling heat pipe

At the point when the writing is reviewed, various researchers have analyzed the air-stream impingement on an intensity sink in calculations, materials, and fly stream speed or spout target distance widely. Ledezma played out a trial, mathematical and hypothetical investigation of the intensity move on a pin-finned plate. They did the connection conditions for ideal blade to-balance dividing and the most extreme warm conductance[24].

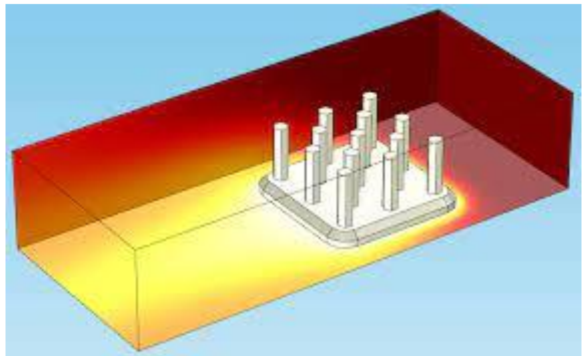


Figure 2 Pinned heat sinks

3. Methodology

It has been determined from past projects that CFD simulation will be used to complete this one. CFD simulation is widely recognized due of its extensive applicability, cost-effectiveness, and control over boundary conditions. A few design concepts came from earlier projects. Figures that follow depict the heat sink model that was previously examined. These models provided guidance on how to create a more effective model, achieving one of the main goals A few somewhat more seasoned investigations use CFD for heat sink reenactments as it were. Linton and Agonafer look at the consequences of itemized CFD displaying of an intensity sink with trial information. Then, at that point, they present a strategy for addressing the intensity sink in a coarse way for less tedious reproductions. Their coarse model concurs well with the nitty gritty model without losing the attributes of the intensity sink. Sathyamurthy and Runstadler concentrated on planar and staggered heat sink execution with financially accessible programming, Familiar. Their computational outcomes concurred well with the trial ones. They observed that the warm execution of staggered balance design is better over planar blade setup. Anyway the tension drop necessity for the amazed blade heat sink was more noteworthy than those for the planar case. This study utilizes CFD for the form heat move reproductions in an entire PC skeleton. The CFD calculations use Icepak and FLUENT simultaneously.. Following steps were followed to accomplish the stages of the project:

Design and CFD analysis of tubed cooling plate

When liquid cooling of component is desired over air cooling in thermal management applications, tubed cold plates from advanced thermal solutions offer a dependable and affordable option. These cold plates have continuous stainless steel or copper tubing. Two base plates are used to secure the stainless steel tubing. Because the copper tubing is attached to the plate directly, rather than via bulky adhesives or thermal insulators, press-fit connection maximizes heat transfer. The 3/8" push-to-connect tubes fittings that are included with Innovative Thermal Technologies Tubed Cold Plates ensure a snug fit between the freezing plate and the inlet and output tubing.

Table 1 Tubed model cooling plate specifications

Plate Material	Aluminum 6063
Tubes material	Copper
Cooling Fluid	Water
Plate size	610×190×15
Tube size	3/8″ outer diameter

a. CFD ANALYSIS OF TUBED COOLING PLATE MODEL

In order to assess the thermal performance of cooling plates, computational fluid dynamics (CFD) analysis simulates the flow of a fluid—typically liquid or air—over and around the plates.

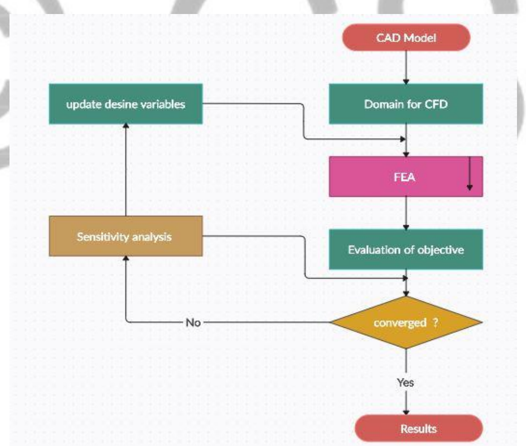


Figure 5 Simulation methods

i. Geometry Preparation:

Make a 3D model with computer-aided design (CAD) software of the plates for cooling and surrounding parts. The morphology that comprises the plates, including the fins, methods, or other elements intended to improve heat transfer, should be faithfully captured in this model.

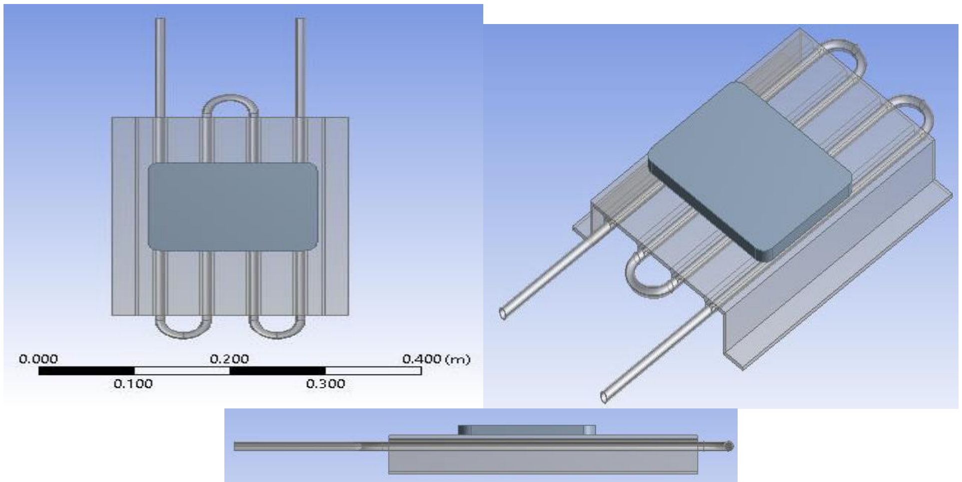


Figure 6 Cad model for conventional cooling plate

ii. Mesh Generation and name selection:

In order to discredit the computational domain, generate a network of edges over the geometry. The mesh should be computationally efficient and sufficient in resolution to capture the significant flow features close to the plates.

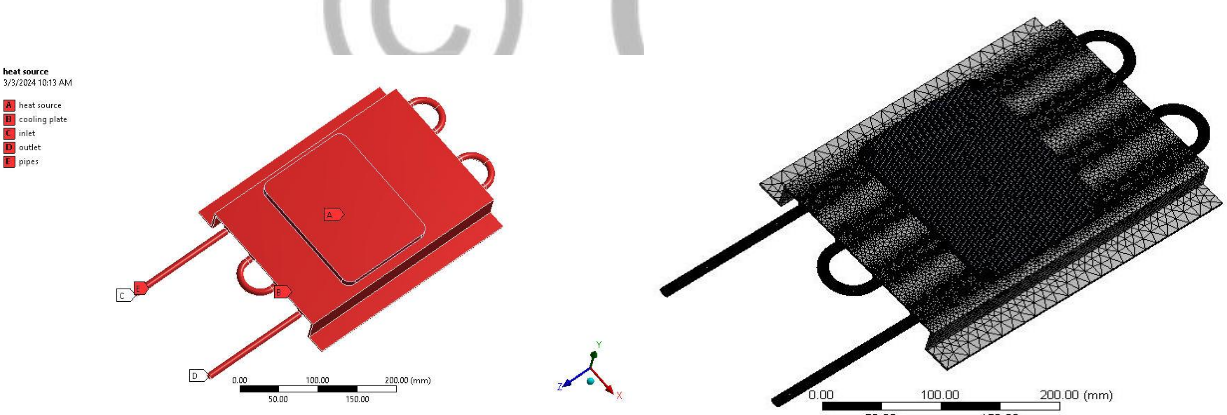


Figure 3 Name selection for cfd Analysis

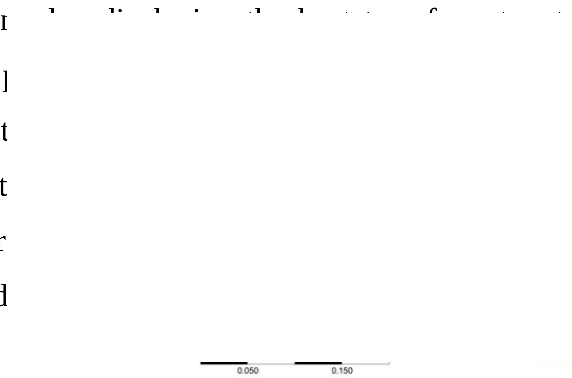
iii. Boundary Conditions:

Establish the CFD simulation's boundary conditions. This entails describing the temperature, entrance velocity, or mass flow rate, and fluid parameters (a density, density, specific heat, etc.) of the fluid approaching the domain. On the cooling plates themselves, boundary conditions could relate to convective heat transfer coefficient, heat flux, or temperature. The aforementioned model was used for the simulations, and the boundary conditions were a velocity of 0.01 m/s and a temperature of 300 K. Heat flux at the intake was estimated to be 320 kW/m2, and flow velocity was assumed to be 0.01 m/s.

4. RESULTS AND DISCUSSIONS

4.1 Temperature drop results with velocity 0.01 m/s

Utilize the post-processing features found in the software for computational fluid dynamics to examine the simulation's outcomes. This could ir mperature contours, and velocity vectors surrounding the cooling ding thermal efficiency, pressure drop, Nusselt number, and heat t oy contrasting them, if available, with experimental data. This cont endability of the simulation forecasts. If necessary, undertake par rified CFD model to enhance the cooling plate design for improved

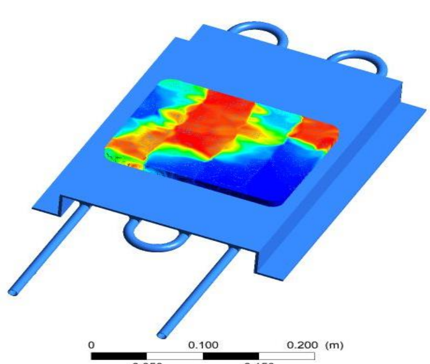


.

4.2 Temperature drop results with velocity 0.02 m/s

A tubed cooling plate was subjected to a computational fluid dynamics, or CFD, simulation in order to assess the thermal performance of the plate under certain heat exchange and flow velocity circumstances. With a temperature gradient of 320 W/m² as well as a flow speed of 0.02 m/s, the study's main objectives were to comprehend the cooling plate's flow behavior and heat transfer properties. Comprehensive insights into the tubed cooling plate's thermal conductivity and flow behavior were obtained from the CFD simulation conducted under conditions of 320 W/m² of heat flux and 0.02 m/s of flow velocity.

Through the analysis regarding temperature transportation, the transfer of heat coefficients, flow features, ranging and thermal endurance metrics, important data was gathered to improve the cooling systems design and functionality. Additional research combining boundary conditions and other characteristics may increase our comprehension of the behavior of the cooling plate and enable advancements in thermal management systems.



4.3 Temperature drop results with velocity 0.03 m/s

Computational Fluid Dynamics (CFD) analysis plays a crucial role in understanding and optimizing the performance of cooling systems, especially those involving high heat fluxes. In this discussion, we'll delve into the CFD analysis of a cooling plate subjected to a heat flux of 320 W/m², with water flowing at a velocity of 0.03 m/s and an inlet temperature (Tin) of 27°C. Cooling plates are commonly used to dissipate heat from electronic components, power systems, and industrial machinery.

The amount of heat emitted from the heating source, in this case 320 W/m², is a significant thermal load, and comprehending how the resulting heat flux affects the ambient temperature the distribution across the surface of the cooling plate is essential for efficient thermal management. The water flow velocity through the cooling processes plate is 0.03 m/s, and this velocity affects the coefficient of convection heat transfer, which determines the efficiency with which heat is transferred from the cooler plate to the flowing fluid. Engineers can optimise the cooling technique's design to maximise heat transfer while minimising energy consumption by analysing the fluid dynamics, including the pattern of flow, turbulence, and pressure drop.

As the water moves through the plate to cool it and take heat from it, its temperature rises. Understanding the temperature distribution within the plate is crucial to maintaining the highest temperature within permissible ranges and preventing the components that are being cooled from overheating.

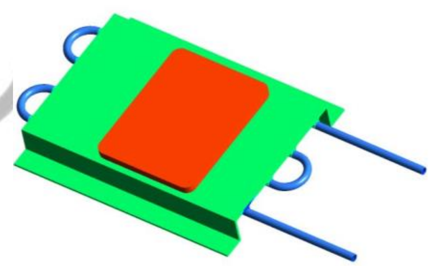
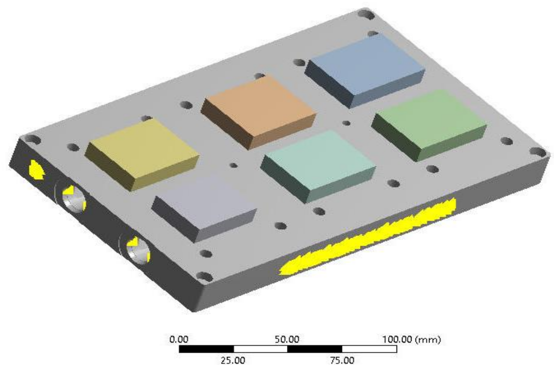


Figure 9 Heat source temperature contour

4.4 Modeling and cfd simulation of Wieland cp 4009d

The goal of this project is to create a tool for single-phase liquid-cooled heat sink design optimisation. The design of the fluid flow is decomposed into flow blocks, which constitute elements arranged in a coarser two-dimensional in nature (2D) grid. Firstly, the functionality of the construction in terms of resistance to thermal expansion, temperature drop, and temperature variation on the surface being heated is obtained by solving the governing energy, mass, and momentum equations for fluid circulation and the transfer of heat within both the liquid and solid domains. Subsequently, the optimal path search (OPS) algorithm is employed to determine the ideal fluid flow arrangement, taking into account the optimisation of the thermal-hydraulic performance criteria. The temperature and fluid flow characteristics inside the Micro Cool heat exchanger's micro channels are simulated by modeling and CFD (computational fluid dynamics) simulation. Here's a broad rundown of the procedure. First, use CAD software such as Solid Works, AutoCAD, or other comparable tools to create a 3D model that represents the The term Wieland Micro Cool heating element shape. Make that the micro channels, which fins, and other components' geometries are appropriately represented in the model. Use meshing tools such as ANSYS The meshing procedure, Gambit, Point by point, or ICEM CFD for producing a computationally mesh for the geometry.



.

Figure 10 Cad model for Wieland cooling plate

In addition to being computationally efficient, the mesh should have enough resolution to properly record the flow dynamics inside the micro channels. Describe the material characteristics of the heat exchanger's components, which are usually aluminum or copper for the micro channels. Give details about the coolant's fluid characteristics (such as density, viscosity, and thermal conductivity) as it passes through the micro channels. The particular use determines which coolant is best. Establish the outlet and inlet boundary conditions, as well as the pressure, temperature, and velocity at the intake. Select a CFD solution that is suitable for managing the intricate circulation of fluid and heat transport processes in the micro channels. ANSYS Fluent, COMSOL programme Multiphysics, Open FOAM to generate and STAR-CCM+ are popular options.

.

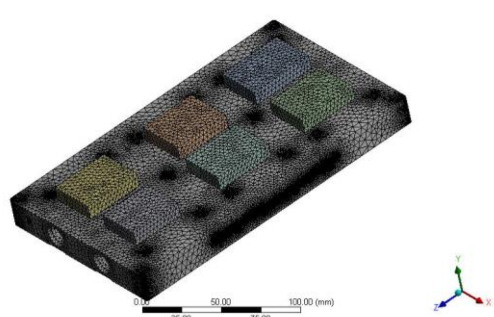


Figure 11 Meshed model

Depending on the intended level of complexity and the application, decide whether a steady-state or transitional simulations are needed. To comprehend the fluid and heat flow behavior within the micro channels, analyse the simulation data. Display pressure distributions, temperatures contours, velocity profiles, and other pertinent data. To assess the efficacy of the heat exchanger design, compute pressure drops, heat transfer coefficients, and other performance measures. To validate the model, compare the outcomes of simulations with available experimental data.

4.5 Simulation Results for Wieland cp 4009d

When it comes to preserving the longevity and optimum performance of electrical products, including processors, effective cooling solutions are critical. Simulations of CFD (computational fluid dynamics) are a vital resource for assessing the performance of cooling techniques. Using a flow frequency of 0.01 m/s and intake temperatures of 27°C, we explore the CFD modeling of a The term Wieland CP4012D conditioning plate that can house six heat-generating computers with a total power consumption of 230,000 W/m² in this discussion. The simulation was painstakingly constructed to mimic actual circumstances.

The principal cooling mechanism is the term Wieland CP4012D conditioning plate, which is well-known for its efficiency and thermal conductivity. The simulation environment faithfully captured the dimensions, material characteristics, and geometric details of the plate. A total of six processors with a combined heat transfer coefficient of 320 W/m² were placed on top of the cooling plate. This setup is similar to what is frequently seen in high-performance computing facilities, where close proximity of numerous processors requires effective thermal control.

4.6 Simulation at velocity of 0.01 m/s

In order to evaluate temperature gradients and possible hotspots, accurate modeling of the heat distribution throughout the plate's surface was made possible by the CFD simulation. Within the simulation domain, a flow frequency of 0.01 m/s was applied in order to replicate the convective transfer of heat mechanism. The extensive study of the dynamics of fluids, encompassing circulation patterns, velocity characteristics, and heat transfer coefficients, is made easier by this low velocity. The simulation sheds light on how the coolant interfaces with the cooling structure and disperses temperature from the processors by combining the fluid circulation equations with the heat transfer equations.

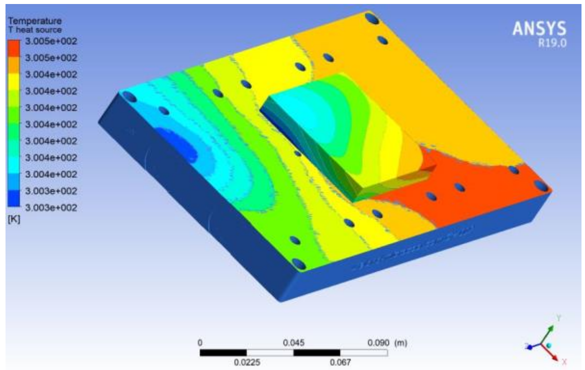


Figure 4 Temperature distribution in heat source

Computational fluid dynamics (CFD) models are examined, with a focus on heat source temperatures dispersion. The simulations employed an average velocity of water of 0.02 m/s and an ambient temperature differential of 320,000 W/m². The results indicate that the temperature range of the heat sources is 300 K to 373 K. The discussion that follows delves into both the reasons and consequences of these temperatures. For every heat source under investigation, the CFD simulations determined a temperature range of 300K to 375 K. The 320 W/m² supplied heat flux has a significant impact on the temperature distribution.

Temperatures near heat sources rise as a result of increased heat flux, expanding the measured range. The water's fixed velocity of 0.02 m/s has an impact on the convective heat transfer coefficient. Speed increases encourage heat transfer through convection, which could cause the temperature near the heat sources to decline. However, the impact of velocity on temperature distribution is complex and dependent on other factors such as shape and flow regime.

.

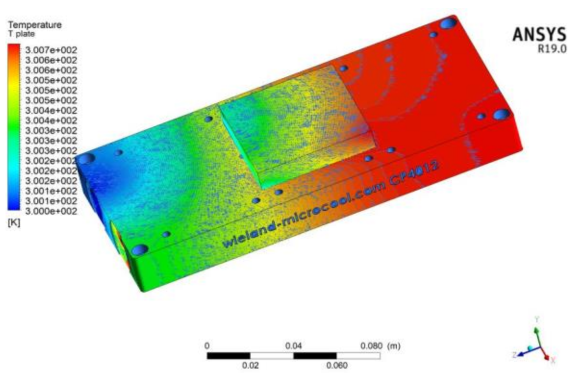


Figure 13 Temperature in pipe 1

4.7 Simulation at velocity of 0.02m/s

The simulation's boundary conditions, which indicate the temp. Of the coolant approaching the system, are set at 27°C for the intake. The temperature distribution over the cooling plate can be seen and examined using CFD analysis. High-temperature regions suggest possible areas of concern and may require additional cooling strategy optimization or cooling plate shape revision.

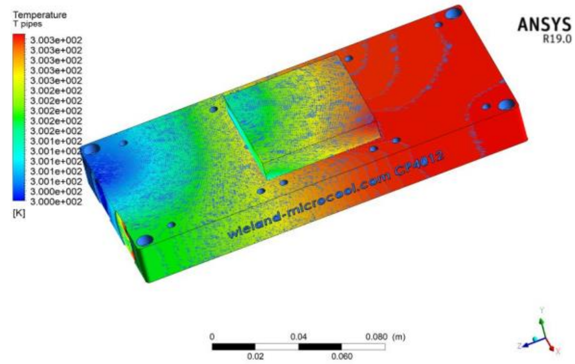


Figure 14 Temperature distribution in top plate

The goal of the computational fluid dynamics, or CFD, study was to evaluate the properties of heat transfer in a system containing sources of heat, a cooling plate, and water acting as the cooling medium. Significant understandings into temperatures distributions, and heat flux, and profiles of velocity were obtained from the analysis, and this knowledge was useful for improving thermal management techniques. Analysing the heat sources' temperature distribution was the main goal of the investigation. A temperature range of 300K to 373K with a heat transfer coefficient of 320 W/m² is shown by the data. Given the strong heat produced by the sources, higher temperatures are to be expected. In order to guarantee operational efficiency and safety, this information is essential for calculating the thermal load on nearby components and for building suitable cooling methods. The water's velocity, which is fixed at 0.02 m/s, also has a big impact on heat dissipation. Convective heat transfer is improved by higher water velocities, which leads to more effective cooling. To provide the best possible system performance, both pressure and velocity drop must be balanced. The research also showed that the cooling plate's temperature increased, peaking at 500K. Given the influence this temperature spike has on the system's overall thermal performance, it warrants attention. Thermal stress from overheating the coolant plate can eventually compromise its foundational strength and functionality.

Therefore, to reduce temperature rise and guarantee the service life of the cooling plate, efficient cooling techniques like improving coolant flow or expanding the heat transfer surface could be required. Additionally, the water coolant's temperature distribution varied throughout the system, from 373K to 500K. This broad temperature range emphasises how crucial it is to comprehend regional differences in cooling effectiveness. Finding hot spots can be useful in locating possible heat accumulation and in directing the installation of extra cooling systems or insulating materials to maximize heat dissipation and avoid overheating.

4.8 Comparative analysis

To direct a near conversation of the outcomes got from miniature cooling plates and Wieland cooling plates, we will dissect the temperature conveyance at different focuses inside the framework. In light of the gave information, we'll examine the distinctions saw between the two sorts of cooling plates. The intensity source temperature is somewhat higher in the Wieland CP 4009 T (300.442°K) contrasted with the miniature cooling plate (300.2k). This distinction could demonstrate a more effective intensity move in the Wieland plate plan. The temperature at the plate zone is likewise marginally higher for the Wieland plate (300.31K contrasted with the miniature cooling plate (300.13K), which could propose better intensity scattering or lower warm obstruction in the Wieland plate. The line temperature is higher in the miniature cooling plate (300.38K) contrasted with the Wieland plate (300.14K). This shows that the miniature cooling plate could encounter higher warm misfortunes along the line.

The bay temperature is comparative for the two plates however somewhat higher in the miniature cooling plate (300.2 K) contrasted with the Wieland plate (300.104K), recommending a possible distinction in stream rate or intensity moves proficiency at the gulf. The power source temperature is essentially higher in the Wieland CP 4009 T (355K) contrasted with the miniature cooling plate (300.2 K). This recommends that the Wieland plate is more powerful at eliminating heat, conceivably because of its plan or material properties. In light of the gave temperature information, the Wieland CP 4009 T seems to display commonly better intensity move execution contrasted with the miniature cooling plate. It keeps up with lower temperatures at basic focuses like the line and outlet, showing more productive cooling. The distinctions noticed could be ascribed to plan varieties in the plates, including surface region, material properties, or stream elements. Further investigation would be expected to completely comprehend the basic variables driving these temperature varieties.

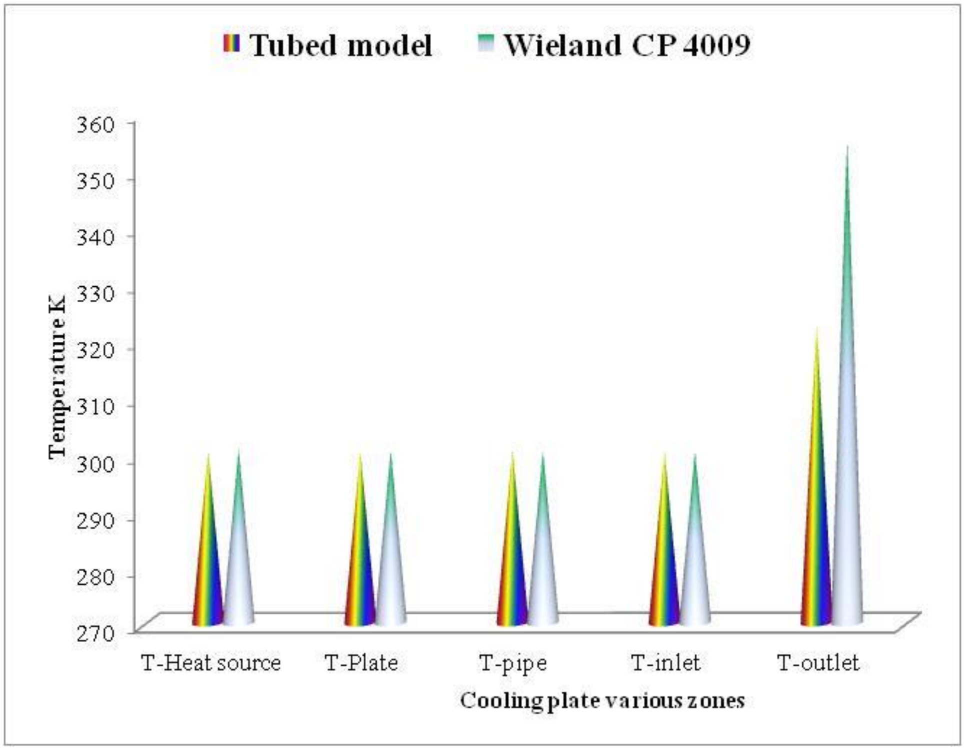


Figure 5 Comparison between proposed plate models

5. CONCLUSION

Fluid cooling can be accomplished in more ways than one, yet the most well known approach is to utilize a plate containing a stream course to move fluid underneath the hardware. The intensity is moved from the plate into the greater system whenever it has been consumed by the fluid. Albeit the most frequently involved liquids in fluid cooling are water or water/glycol, extra liquids that can be utilized incorporate petroleum, oil, and refrigerant. It still up in the air from past undertakings that CFD recreation will be utilized to finish this one. CFD reenactment is generally perceived due of its broad relevance, cost-viability, and command over limit conditions. A couple of plan ideas came from before projects. Figures that follow portray the intensity sink model that was recently inspected. A tubed cooling plate was exposed to a computational liquid elements, or CFD, reenactment to survey the warm presentation of the plate under specific intensity trade and stream speed conditions. With a temperature inclination of 320 W/m² as well as a stream speed of 0.02 m/s, the review's principal goals were to fathom the cooling plate's stream conduct and intensity move properties. Thorough experiences into the tubed cooling plate's warm conductivity and stream conduct were gotten from the CFD recreation directed under states of 320 kW/m² of intensity transition and 0.02 m/s of stream speed. The intensity source temperature is marginally higher in the Wieland CP 4009 T (300.442°K) contrasted with the miniature cooling plate (300.2 K). This distinction could show a more proficient intensity move in the Wieland plate plan. The temperature at the plate zone is additionally marginally higher for the Wieland plate (300.31K) contrasted with the miniature cooling plate (300.13K), which could propose better intensity dispersal or lower warm obstruction in the Wieland plate. The line temperature is higher in the miniature cooling plate (300.38K) contrasted with the Wieland plate (300.14K). This demonstrates that the miniature cooling plate could encounter higher warm misfortunes along the line.

The delta temperature is comparative for the two plates yet somewhat higher in the miniature cooling plate (300.2 K) contrasted with the Wieland plate (300.104K), recommending an expected distinction in stream rate or intensity moves effectiveness at the bay. The power source temperature is altogether higher in the Wieland CP 4009 T (355K) contrasted with the miniature cooling plate (300.2 K). This proposes that the Wieland plate is more successful at eliminating heat, potentially because of its plan or material properties. In light of the gave temperature information, the Wieland CP 4009 T seems to display commonly better intensity move execution contrasted with the miniature cooling plate. It keeps up with lower temperatures at basic focuses like the line and outlet, demonstrating more productive cooling

References

[1] W. Zuo, D. Li, Q. Li, Q. Cheng, and Y. Huang, "Effects of intermittent pulsating flow on the performance of multi-channel cold plate in electric vehicle lithium-ion battery pack," *Energy*, vol. 294, p. 130832, 2024.

[2] Z. Zou, J. Xie, Y. Luo, G. Zhang, and X. Yang, "Numerical study on a novel thermal management system coupling immersion cooling with cooling tubes for power battery modules," *J. Energy Storage*, vol. 83, p. 110634, 2024.

[3] J. Zheng *et al.*, "A novel thermal management system combining phase change material with wavy cold plate for lithium-ion battery pack under high ambient temperature and rapid discharging," *Appl. Therm. Eng.*, vol. 245, p. 122803, 2024.

[4] J. Yuan *et al.*, "Structure optimization design and performance analysis of liquid cooling plate for power battery," *J. Energy Storage*, vol. 87, p. 111517, 2024.

[5] M. Yang, G. Mathew, H. Nemati, and M. A. Moghimi, "A novel approach for active cooling of a battery at cell level: Air-cooled mini-channel heat sink, enhanced with intermittent metal foam," *J. Energy Storage*, vol. 81, p. 110374, 2024.

[6] J. Xun, R. Liu, and K. Jiao, "Numerical and analytical modeling of lithium ion battery thermal behaviors with different cooling designs," *J. Power Sources*, vol. 233, pp. 47–61, 2013.

[7] Võsa, Karl-Villem, Ferrantelli, Andrea, and Kurnitski, Jarek, "Experimental study of radiator, underfloor, ceiling and air heater systems heat emission performance in TUT nZEB test facility," *E3S Web Conf.*, vol. 111, p. 4005, 2019.

[8] Võsa, Karl-Villem, Ferrantelli, Andrea, and Kurnitski, Jarek, "Annual performance analysis of heat emission in radiator and underfloor heating systems in the European reference room.," *E3S Web Conf.*, vol. 111, p. 4009, 2019.

[9] A. Vadiee, A. Dodoo, and E. Jalilzadehazhari, "Heat Supply Comparison in a Single-Family House with Radiator and Floor Heating Systems," *Buildings*, vol. 10, no. 1, 2020.

[10] S. F. Tie and C. W. Tan, "A review of energy sources and energy management system in electric vehicles," *Renew. Sustain. Energy Rev.*, vol. 20, pp. 82–102, 2013.

[11] Z. Tian, Y. Zhou, Z. Cao, and W. Gao, "Microchannel cooling plates with non-uniform airfoil fin arrangement for large-capacity marine battery: An experimental study," *Appl. Therm. Eng.*, vol. 241, p. 122393, 2024.

[12] B. W. Olesen and M. de Carli, "Calculation of the yearly energy performance of heating systems based on the European Building Energy Directive and related CEN standards," *Energy Build.*, vol. 43, no. 5, pp. 1040–1050, 2011.

[13] M. Maivel and J. Kurnitski, "Radiator and floor heating operative temperature and temperature variation corrections for EN 15316-2 heat emission standard," *Energy Build.*, vol. 99, pp. 204–213, 2015.

[14] Y. Huo, Z. Rao, X. Liu, and J. Zhao, "Investigation of power battery thermal management by using mini-channel cold plate," *Energy Convers. Manag.*, vol. 89, pp. 387–395, 2015.

[15] A. Hesaraki and S. Holmberg, "Energy performance of low temperature heating systems in five new-built Swedish dwellings: A case study using simulations and on-site measurements," *Build. Environ.*, vol. 64, pp. 85–93, 2013.

[16] Y. Chen, K. Chen, Y. Dong, and X. Wu, "Bidirectional symmetrical parallel mini-channel cold plate for energy efficient cooling of large battery packs," *Energy*, vol. 242, p. 122553, 2022.

[17] T. Amalesh and N. L. Narasimhan, "Introducing new designs of minichannel cold plates for the cooling of Lithium-ion batteries," *J. Power Sources*, vol. 479, p. 228775, 2020.

[18] O. Bayer, R. Oskay, A. Paksoy, and S. Aradag, "CFD simulations and reduced order modeling of a refrigerator compartment including radiation effects," *Energy Convers. Manag.*, vol. 69, pp. 68–76, 2013.

[19] J. M. Belman-Flores and A. Gallegos-Muñoz, "Analysis of the flow and temperature distribution inside the compartment of a small refrigerator," *Appl. Therm. Eng.*, vol. 106, pp. 743–752, 2016.

[20] J. M. Belman-Flores, A. Gallegos-Muñoz, and A. Puente-Delgado, "Analysis of the temperature stratification of a no-frost domestic refrigerator with bottom mount configuration," *Appl. Therm. Eng.*, vol. 65, no. 1, pp. 299–307, 2014.

[21] K. Chen, Z. Zhang, B. Wu, M. Song, and X. Wu, "An air-cooled system with a control strategy for efficient battery thermal management," *Appl. Therm. Eng.*, vol. 236, p. 121578, 2024.

[22] V. G. Choudhari, A. S. Dhoble, and S. Panchal, "Numerical analysis of different fin structures in phase change material module for battery thermal management system and its optimization," *Int. J. Heat Mass Transf.*, vol. 163, p. 120434, 2020.

[23] T. Deng, Y. Ran, G. Zhang, and Y. Yin, "Novel leaf-like channels for cooling rectangular lithium ion batteries," *Appl. Therm. Eng.*, vol. 150, pp. 1186–1196, 2019.

[24] K. Fukuyo, T. Tanaami, and H. Ashida, "Thermal uniformity and rapid cooling inside refrigerators," *Int. J. Refrig.*, vol. 26, no. 2, pp. 249–255, 2003.

[25] J. L. Junfei Zhou Xinjun Wang and H. Lu, "CFD analysis of mist/air film cooling on a flat plate with different hole types," *Numer. Heat Transf. Part A Appl.*, vol. 71, no. 11, pp. 1123–1140, 2017.

[26] Y. Li, Z. Zhou, and W.-T. Wu, "Three-dimensional thermal modeling of Li-ion battery cell and 50 V Li-ion battery pack cooled by mini-channel cold plate," *Appl. Therm. Eng.*, vol. 147, pp. 829–840, 2019.

[27] Z. Liu, Y. Wang, J. Zhang, and Z. Liu, "Shortcut computation for the thermal management of a large air-cooled battery pack," *Appl. Therm. Eng.*, vol. 66, no. 1, pp. 445–452, 2014.

[28] L. Ming, W. Jianchao, G. Qin, L. Yue, X. Qingfeng, and Q. Guihe, "Numerical Analysis of Cooling Plates with Different Structures for Electric Vehicle Battery Thermal Management Systems," *J. Energy*

Eng., vol. 146, no. 4, p. 4020037, Aug. 2020.

[29] E. Söylemez, E. Alpman, A. Onat, and S. Hartomacıoğlu, "CFD analysis for predicting cooling time of a domestic refrigerator with thermoelectric cooling system," *Int. J. Refrig.*, vol. 123, pp. 138–149, 2021.

[30] K. Chen, M. Song, W. Wei, and S. Wang, "Structure optimization of parallel air-cooled battery thermal management system with U-type flow for cooling efficiency improvement," *Energy*, vol. 145, pp. 603–613, 2018.

[31] L. Hongkun, M. M. Noor, Y. Wenlin, K. Kadirgama, I. A. Badruddin, and S. Kamangar, "Experimental research on heat transfer characteristics of a battery liquid-cooling system with ⊥-shaped oscillating heat pipe under pulsating flow," *Int. J. Heat Mass Transf.*, vol. 224, p. 125363, 2024.

[32] W. Jiang, P. Lyu, X. Liu, and Z. Rao, "An immersion flow boiling heat dissipation strategy for efficient battery thermal management in non-steady conditions," *Appl. Therm. Eng.*, vol. 245, p. 122783, 2024.

[33] Y. Lai, W. Wu, K. Chen, S. Wang, and C. Xin, "A compact and lightweight liquid-cooled thermal management solution for cylindrical lithium-ion power battery pack," *Int. J. Heat Mass Transf.*, vol. 144, p. 118581, 2019.

[34] H. Park, "A design of air flow configuration for cooling lithium ion battery in hybrid electric vehicles," *J. Power Sources*, vol. 239, pp. 30–36, 2013.

[35] D. Yu, W. Huang, X. Wan, S. Fan, and T. Sun, "Optimization of simultaneous utilization of air and water flow in a hybrid cooling system for thermal management of a lithium-ion battery pack," *Renew. Energy*, vol. 225, p. 120248, 2024.