ReGeo ISSN: 2177-3246

EXPERIMENTAL AND NUMERICAL INVESTIGATION OF NATURAL CONVECTION IN PLATE-FIN HEAT SINKS USING COMSOL® MULTIPHYSICS

INVESTIGAÇÃO EXPERIMENTAL E NUMÉRICA DA CONVECÇÃO NATURAL EM DISSIPADORES DE CALOR A ALETAS PLANAS UTILIZANDO O COMSOL® MULTIPHYSICS

INVESTIGACIÓN EXPERIMENTAL Y NUMÉRICA DE LA CONVECCIÓN NATURAL EN DISIPADORES DE CALOR DE ALETAS PLANAS MEDIANTE COMSOL® MULTIPHYSICS

https://doi.org/10.56238/revgeov17n11-039

Data de submissão: 06/10/2026 **Data de publicação:** 06/11/2026

Vilson Altair da Silva[1], Ana Lúcia Fernandes de Lima e Silva[2], Sandro Metrevelle Marcondes de Lima e Silva[3], Rogério Fernandes Brito[4]

ABSTRACT

Heat sinks are commonly used to enhance heat transfer and maintain suitable operating temperatures in electronic and other heat-generating devices. In this study, natural convection heat transfer under transient and steady-state conditions was investigated in an aluminum heat sink with flat rectangular fins. Experimental tests and numerical simulations using COMSOL® Multiphysics 5.2 were performed to evaluate the temperature distribution and to determine an appropriate positioning of the temperature sensors. The computational domain and boundary conditions were defined to reproduce the experimental setup, and the heat flux generated by a resistive heater was imposed as the thermal input to the numerical model. Experimental and numerical results were compared using the steady-state temperature measured at the fin tip. The numerical model showed good agreement with the experimental data, with a temperature difference of approximately 1.3 K for the selected mesh, supporting the use of the computational model for the analysis of natural convection heat transfer in the heat sink.

Keywords: Heat Sinks. Natural Convection. Experimental and Numerical Studies. COMSOL®.

RESUMO

Dissipadores de calor são comumente utilizados para melhorar a transferência de calor e manter temperaturas operacionais adequadas em dispositivos eletrônicos e outros equipamentos que geram calor. Neste estudo, investigou-se a transferência de calor por convecção natural, tanto em condições transientes quanto em regime permanente, em um dissipador de calor de alumínio com aletas retangulares planas. Foram realizados testes experimentais e simulações numéricas, utilizando o software COMSOL® Multiphysics 5.2, para avaliar a distribuição de temperatura e determinar o posicionamento adequado dos

[1] Instituto de Engenharia Mecânica (IEM) - Campus Prof. José Rodrigues Seabra - Sede, Universidade Federal de Itajubá (UNIFEI). Itajubá, MG, Brazil. E-mail: altairwayne@gmail.com
[2] Instituto de Engenharia Mecânica (IEM) - Campus Prof. José Rodrigues Seabra - Sede, Universidade Federal de Itajubá (UNIFEI). Itajubá, MG, Brazil. E-mail: alfsilva@unifei.edu.br,
[3] Instituto de Engenharia Mecânica (IEM) - Campus Prof. José Rodrigues Seabra - Sede, Universidade Federal de Itajubá (UNIFEI). Itajubá, MG, Brazil. E-mail: metrevel@unifei.edu.br
[4] Instituto de Engenharia Integrada (IEI) - Campus Theodomiro Carneiro Santiago - Itabira, Universidade Federal de Itajubá (UNIFEI). Itabira, MG, Brazil. E-mail: rogbrito@unifei.edu.br

sensores de temperatura. O domínio computacional e as condições de contorno foram definidos de modo a reproduzir a configuração experimental, sendo o fluxo de calor gerado por um aquecedor resistivo estabelecido como a entrada térmica para o modelo numérico. Os resultados experimentais e numéricos foram comparados com base na temperatura em regime permanente medida na extremidade da aleta. O modelo numérico apresentou boa concordância com os dados experimentais, com uma diferença de temperatura de aproximadamente 1,3 K para a malha selecionada, o que valida a utilização do modelo computacional para a análise da transferência de calor por convecção natural no dissipador de calor.

Palavras-chave: Dissipadores de Calor. Convecção Natural. Estudos Experimentais e Numéricos. COMSOL®.

RESUMEN

Los disipadores de calor se utilizan habitualmente para mejorar la transferencia de calor y mantener temperaturas de funcionamiento adecuadas en dispositivos electrónicos y otros equipos que generan calor. En este estudio, se investigó la transferencia de calor por convección natural, tanto en condiciones transitorias como en estado estacionario, en un disipador de calor de aluminio con aletas rectangulares planas. Se llevaron a cabo ensayos experimentales y simulaciones numéricas mediante COMSOL® Multiphysics 5.2 con el fin de evaluar la distribución de temperatura y determinar la ubicación óptima de los sensores de temperatura. El dominio computacional y las condiciones de contorno se definieron para reproducir la configuración experimental, estableciéndose el flujo de calor generado por una resistencia calefactora como entrada térmica para el modelo numérico. Los resultados experimentales y numéricos se compararon utilizando la temperatura en estado estacionario medida en el extremo de la aleta. El modelo numérico mostró una buena concordancia con los datos experimentales —con una diferencia de temperatura de aproximadamente 1,3 K para la malla seleccionada—, lo que respalda el uso de este modelo computacional para el análisis de la transferencia de calor por convección natural en el disipador.

Palabras clave: Disipadores de Calor. Convección Natural. Estudios Experimentales y Numéricos. COMSOL®.

ISSN: 2177-3246

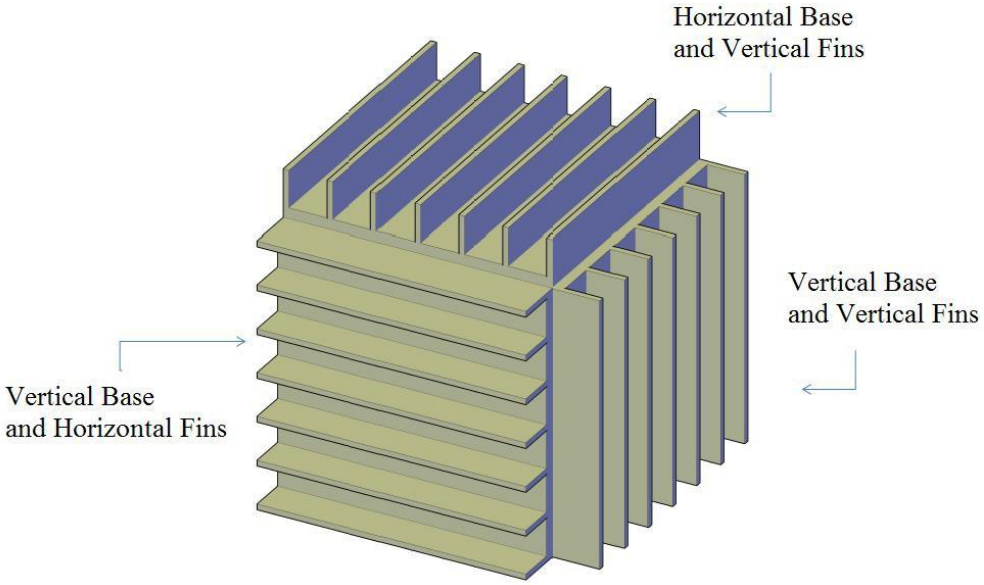
1 INTRODUCTION

Fins are extended surfaces used to enhance heat transfer between a surface and the surrounding fluid. The use of finned surfaces increases the heat transfer rate by increasing the surface area available for heat transfer. Fins are widely used in equipment and devices that generate heat during operation, such as motors, transformers, and microprocessors, particularly when heat must be dissipated within limited spaces.

In Figure 1, taken from the article by Leung, Probert and Shilston (1985), illustrates the positions of the heatsinks.

Figure 1

Relative positions of the base and the fins of the heat sink.



Source: Leung, Probert and Shilston (1985).

Numerical simulations were carried out by Tari and Mehrtash (2013) to investigate the effect of heat sink orientation on the Nusselt number. The direction of gravitational acceleration relative to the heat sink was varied by changing the inclination angle of the heat sink base. The authors proposed correlations for the Nusselt number as a function of the inclination angle for two ranges of a modified Rayleigh number. Streamline patterns obtained from the numerical simulations were also presented, providing further insight into the natural convection flow around the heat sink.

Kim et al. (2013) experimentally investigated natural convection from vertically oriented plate-fin heat sinks and used FLUENT® to examine the flow and heat transfer characteristics. The correlation proposed by the authors was applicable over a wider range of Rayleigh

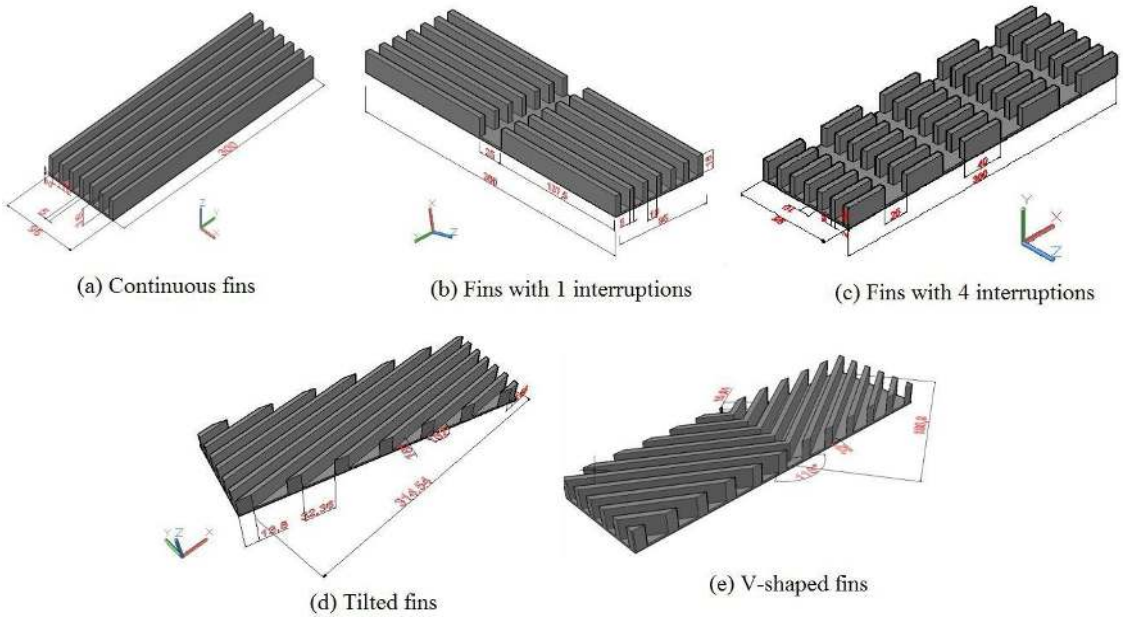
numbers and was reported to be more accurate than some previously proposed correlations. Based on an asymptotic approach, the authors showed that the optimal fin spacing is independent of fin height but depends on fin length, the temperature difference between the heat sink and the surrounding environment, and the thermophysical properties of the fluid.

Shen et al. (2014) investigated the effect of orientation on the thermal performance of a heat sink for LED applications. The heat sink was experimentally tested at eight different angular positions at 45° intervals. The lowest thermal performance was observed at an orientation angle of 270°, whereas better performance was obtained at 225° and 315°, particularly at 315°. The computational results showed that the mismatch between the available heat transfer area and the natural convection flow, together with flow blockage, were the two dominant factors responsible for the deterioration of heat transfer in rectangular-fin heat sinks.

Khudheyer and Hasan (2015) investigated the effects of different fin configurations and heat fluxes through experimental tests and numerical simulations performed using COMSOL® 5.0. Five heat sink configurations were considered: continuous fins, fins with one interruption, fins with four interruptions, inclined fins, and V-shaped fins, as shown in Fig. 2. The heat sink with four interruptions exhibited the largest temperature difference between the base and the surrounding environment, whereas the configuration with one interruption exhibited the smallest difference. The authors also observed that the heat transfer coefficient increased with fin height. For fins with a fixed length of 300 mm and a height of 18 mm, varying the fin spacing resulted in an optimal spacing of 10 mm. For fins with a fixed height of 18 mm and spacing of 5 mm, the heat transfer coefficient reached its lowest value at a fin length of 500 mm for a fixed heat flux. The influence of the heat sink base thickness was also investigated, and the heat transfer coefficient reached its lowest value for a base thickness of 9 mm when the fin height and spacing were 18 mm and 5 mm, respectively.

Figure 2

Heat sinks used by Khudheyer and Hasan (2015).



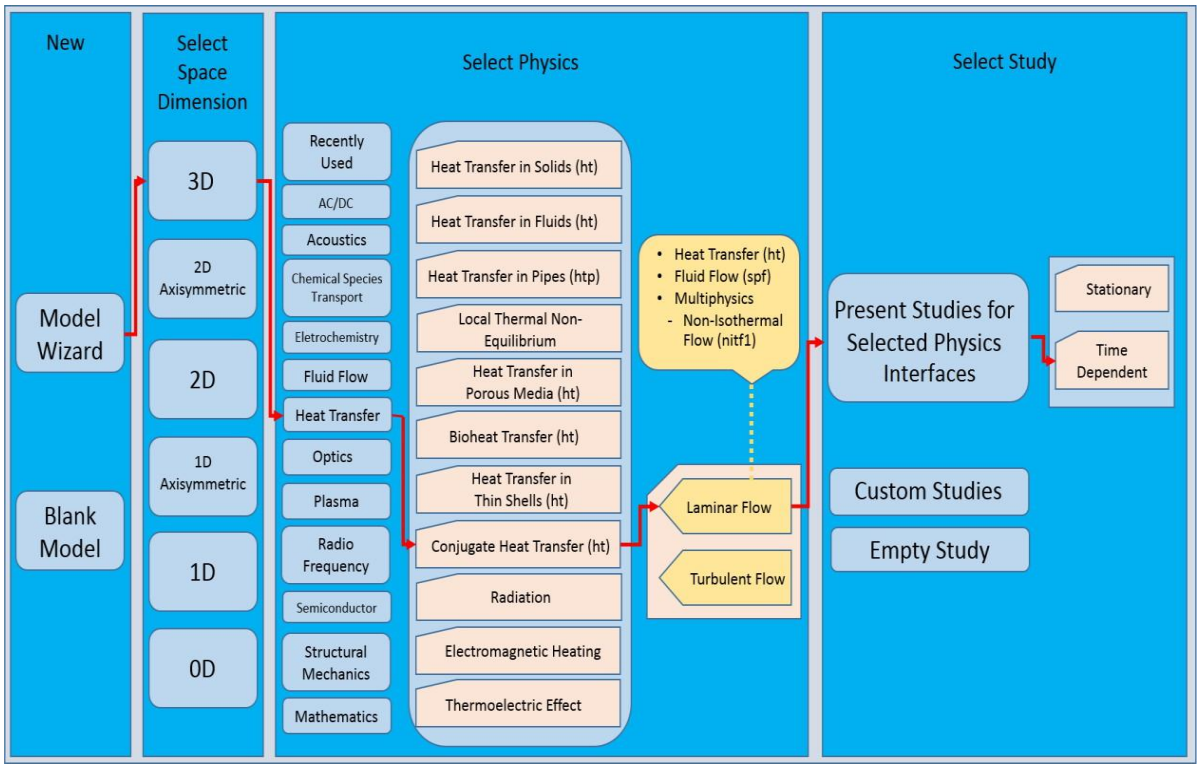
Ong et al. (2017) investigated thermal spreading effects and determined the associated thermal spreading resistance. The performance of two conventional heat sinks, consisting of a flat metal base with an array of fins, was evaluated under natural and forced convection for different heat inputs. The numerical simulations showed that the temperature distribution over the heat sink was nonuniform, indicating that thermal spreading resistance can adversely affect heat transfer performance. The effect of thermal spreading was found to be considerably less pronounced under forced convection.

2 METHODOLOGY

Numerical simulations were performed using COMSOL Multiphysics 5.2. The numerical model couples heat conduction in the solid domain, heat transfer in the fluid domain, and buoyancy-driven fluid flow resulting from temperature-induced density variations. This multiphysics coupling results in a system of equations that must be solved simultaneously. Figure 3 summarizes the procedure used to configure the numerical model in COMSOL Multiphysics 5.2, including the spatial dimension, the physics interfaces, and the type of study (transient or steady-state).

Figure 3

Organization chart of a COMSOL problem.



The numerical model was developed to reproduce the experimental setup, which consisted of a wooden support, an MDF board used for thermal insulation, a resistive heater, and an aluminum heat sink. The experimental setup is shown in Fig. 4.

Figure 4

Arrangement of test bench.

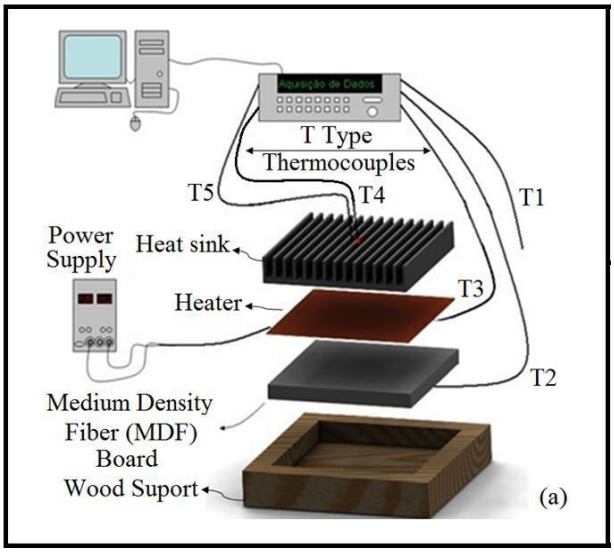
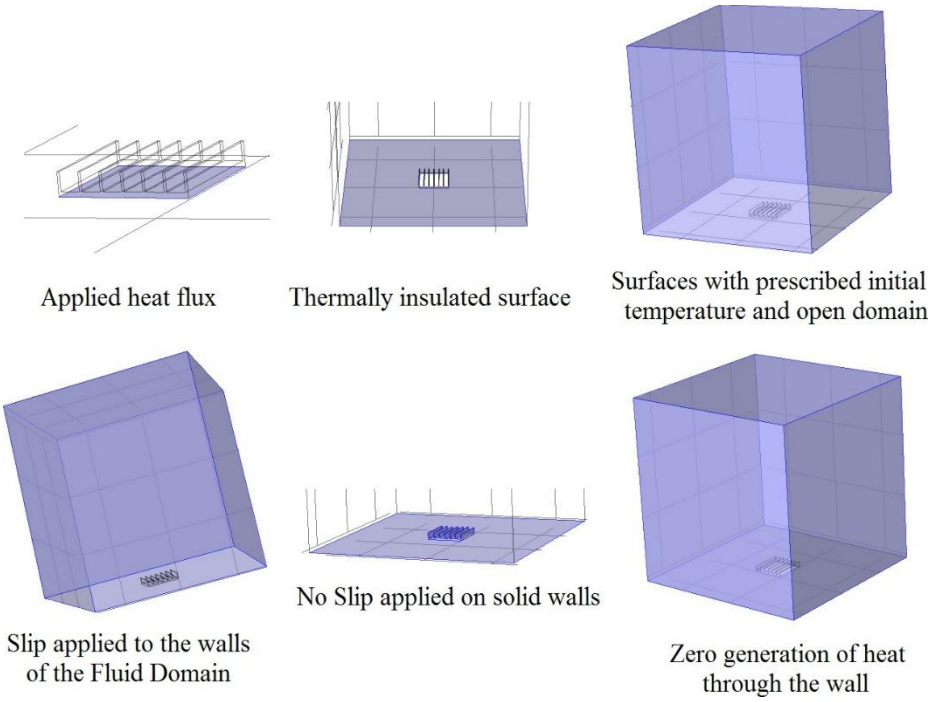


Figure 5 shows the boundary conditions used in the computational model. These boundary conditions were defined to reproduce the conditions of the experimental tests.

Figure 5

Boundary Conditions.



3 RESULTS

Tests were performed using thermocouples welded to the fin tips at different positions along the heat sink to evaluate the temperature distribution. Figure 6(a) shows the thermocouple locations for the horizontally oriented heat sink. The results shown in Fig. 6(b) indicate an approximately uniform temperature distribution, with negligible temperature differences among the measurement positions. Therefore, only the thermocouple located at the central position was used for subsequent temperature measurements, as shown in Fig. 6(a2).

Figure 6

(a1) Thermocouples positioning for the test of uniformity of temperature; (a2) position used in the other tests; (b) temperature distribution to the test of uniformity.

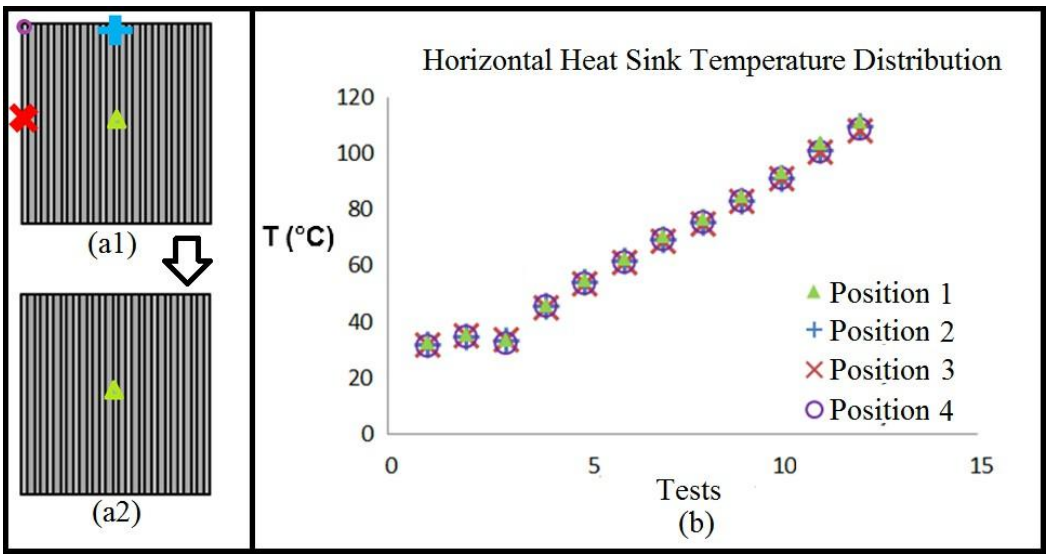


Figure 7 shows the temperature distribution over the heat sink obtained from the COMSOL® simulation. The numerical results indicate a small temperature variation between the central region and the fin tip, consistent with the experimental results shown in Fig. 6(b). Therefore, both the experimental and numerical results indicate an approximately uniform temperature distribution within the heat sink under steady-state conditions.

Figure 7

Isotherms obtained by COMSOL® (a) from the top view of the heatsink, (b) the outermost fin of the heat sink, (c) from the front view of the heat sink.

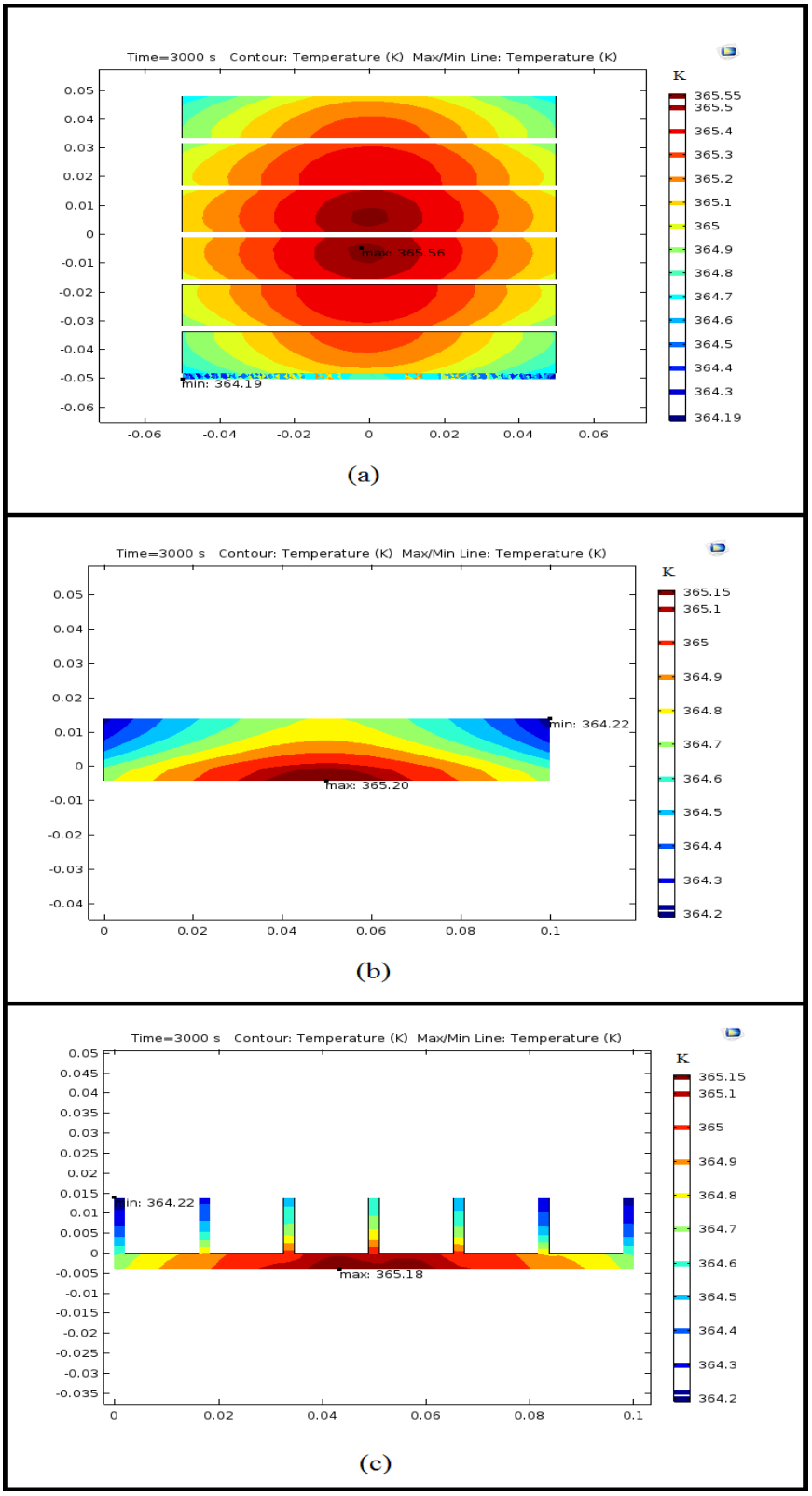


Figure 8 shows some of the meshes used in the simulations, and Table 1 presents the corresponding number of elements. COMSOL® provides automatic mesh generation with different options for mesh refinement and element configuration. A tetrahedral mesh was selected for the subsequent simulations of the inclined heat sink, with the mesh parameters adjusted according to the requirements of the computational domains.

Figure 8

Meshes used in the simulations of Cases 3, 4, and 5 in (a) a complete view and (b) an enlarged view of the heat sink.

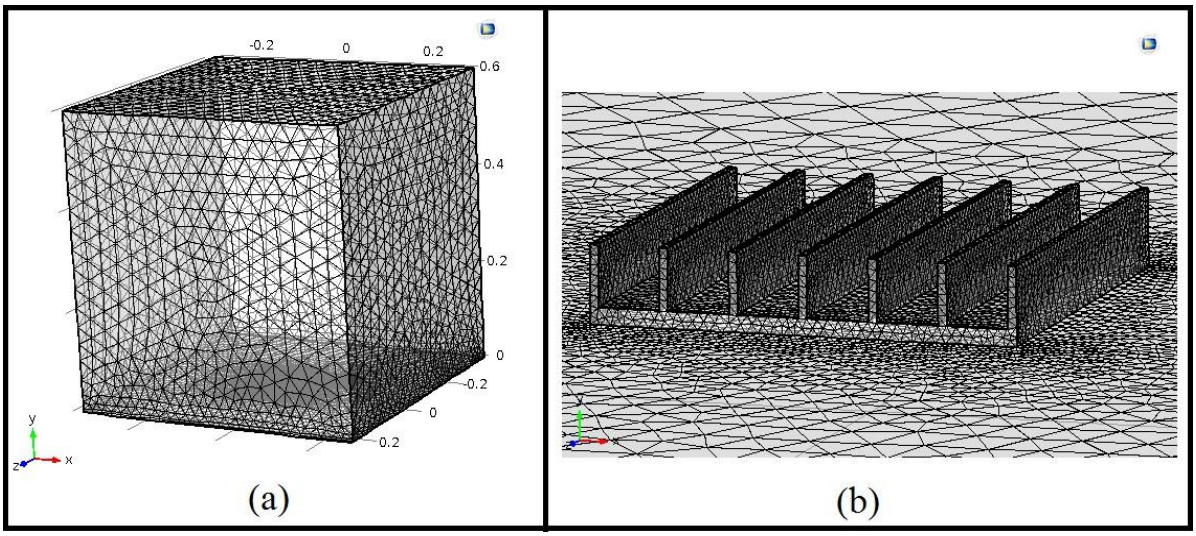


Table 1 summarizes the mesh refinement cases considered in the simulations, including the number of elements and the corresponding computational time. All cases simulated the same experimental conditions, with only the mesh refinement level being varied. These cases were compared to support the selection of an appropriate mesh for the subsequent simulations.

Table 1

Mesh test and time to reach the steady state.

Simulated Case	Total number of Tetrahedral Elements	Temperature with 40% of Simulation Performed [K]	Temperature [K]	Simulation Time
Case 1	84,402	349.31	353.29	20min 39s
Case 2	153,821	354.14	359.69	7h 34min 9s
Case 3	411,135	357.90	365.28	31h 19min 11s
Case 4	411,135	357.85	365.40	41h 27min 45s
Case 5	411,135	357.92	365.67	62h 51min 47s
Case 6	2,128,255	358.76	***	2 weeks to run 40% of simulation

The temperatures reported in Table 1 were obtained at the same location as thermocouple T4 in the experimental setup, as shown in Fig. 5(a2). The same heat flux used in the experimental tests was imposed in the numerical simulations, allowing the experimental and numerical temperatures to be compared. The experimental steady-state temperature was 366.97 K, whereas Case 5 yielded a numerical temperature of 365.67 K, corresponding to a difference of approximately 1.3 K.

The comparison indicates good agreement between the experimental and numerical results for Case 5. Figure 9 shows the temperature evolution over 6000 s for both the experimental test and the numerical simulations, allowing the time required to reach steady-state conditions to be evaluated. To reduce the computational cost, Cases 3 and 4 were simulated for shorter periods of 3,000 and 4,000 s, respectively, to determine the minimum simulation time required to reach steady-state conditions.

Figure 9

Temperature curves until the steady state.

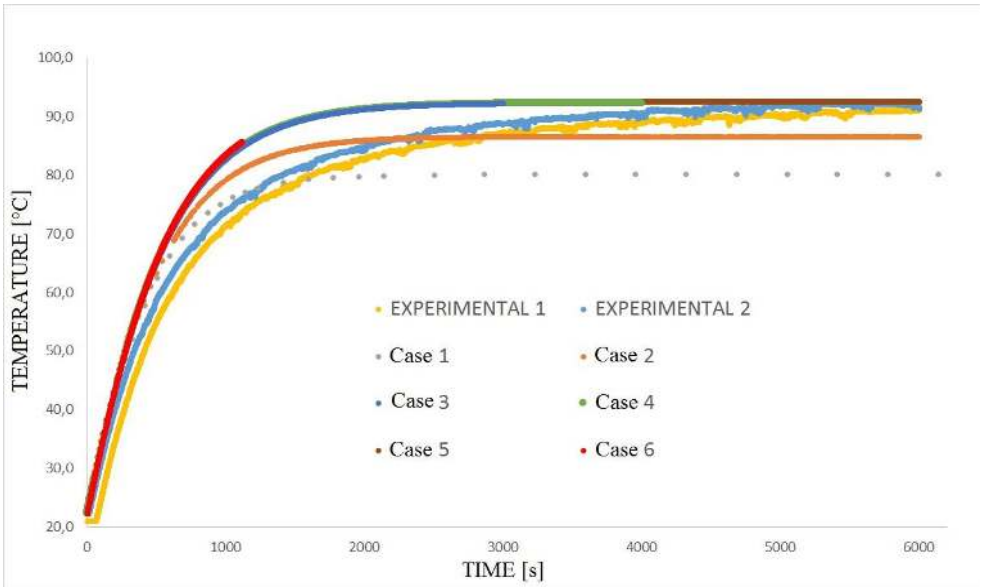
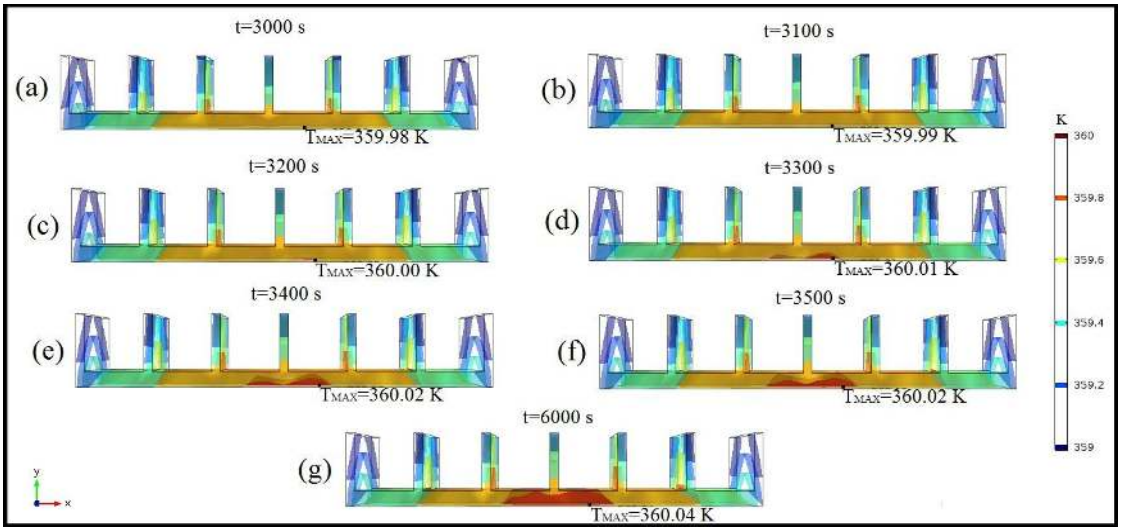


Figure 10 shows the temperature distribution in the heat sink at different times to determine the minimum simulation time required to reach steady-state conditions. Following Kim et al. (2011), steady-state conditions were assumed to be reached when the temperature variation at the bottom of the heat sink did not exceed 0.1 °C over a 2-min period.

Figure 10

Temperature distribution on the heat sinks in (a) 3000 seconds, (b) 3100 seconds, (c) 3200 seconds, (d) 3300 seconds, (e) 3400 seconds, (f) 3500 seconds, and (g) 6000 seconds.



After 3000 s (50 min), the variation in the maximum heat sink temperature did not exceed 0.06 °C. Therefore, the heat sink was considered to have reached steady-state conditions after approximately 3000 s.

Figure 11 shows the temporal evolution of the temperature field throughout the computational domain, together with enlarged views of the heat sink at different time instants. As the heat sink temperature increases, a thermal plume develops above the heat sink and rises due to buoyancy. The upward motion of the heated air promotes convective heat transfer from the heat sink to the surrounding fluid. Air adjacent to the heat sink is heated, causing its density to decrease. The warmer, less dense air rises and is replaced by cooler, denser air from the surroundings, thereby sustaining the natural convection process.

Figure 11

Time evolution of the temperature field in an XY plane in the center of the domain at the instant (a) and (b) 100 seconds, (c) and (d) 1000 seconds, (e) and (f) 3000 seconds, and (g) and (h) 6000 seconds.

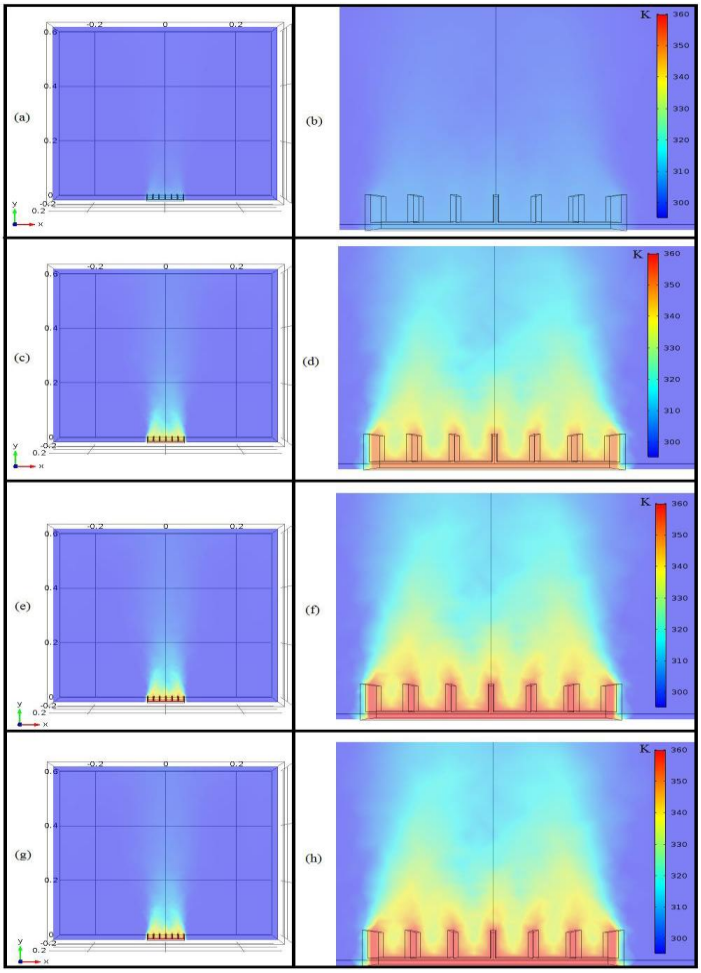


Figure 10 shows that only a negligible temperature variation occurred between 3000 and 6000 s, indicating that steady-state conditions had been reached by approximately 3000 s

Table 2 compares the temperatures obtained from the numerical simulations with the experimental temperature. The experimental steady-state temperature was 366.97 K. Among the completed simulations, Case 5 showed the closest agreement with the experimental result, with a temperature difference of 1.30 K, corresponding to a percent difference of 0.35%

Table 2

Comparison between simulated and experimental temperatures.

Simulated Case	Temperature [K]	ΔT Between Simulation and Experiments [K]	Percent Difference [%]
Case 1	353.29	13.68	3.73
Case 2	359.69	7.28	1.98
Case 3	365.28	1.69	0.46
Case 4	365.40	1.57	0.43
Case 5	365.67	1.30	0.35
Case 6	***	***	***

4 CONCLUSION

The experimental and numerical investigation demonstrated that COMSOL® Multiphysics 5.2 can adequately reproduce the thermal behavior of the heat sink under natural convection for the conditions investigated. The transient analysis showed that steady-state conditions were reached after approximately 3000 s. Beyond this time, the variation in the maximum heat sink temperature remained below 0.06 °C, indicating that a simulation time of approximately 3000 s is sufficient to represent the steady-state thermal response while reducing the computational cost.

The comparison between the experimental and numerical results showed good agreement for the selected mesh. The experimental steady-state temperature was 366.97 K, whereas Case 5 yielded a numerical temperature of 365.67 K, corresponding to a difference of 1.30 K, or 0.35%. In addition, the experimental measurements and numerical temperature field indicated an approximately uniform temperature distribution at the evaluated locations of the heat sink under steady-state conditions.

The numerical model also provided information that is difficult to obtain experimentally, particularly the temporal evolution of the temperature field in the surrounding air. The

simulations allowed the development of the thermal plume above the heat sink to be visualized, providing further insight into the buoyancy-driven flow responsible for natural convection heat transfer.

Overall, the results support the use of the proposed numerical model for the analysis of natural convection heat transfer in plate-fin heat sinks. Accurate representation of the experimental geometry and boundary conditions was essential for obtaining agreement between the numerical and experimental results. Future work should investigate the influence of mesh element type and computational-domain dimensions and extend the numerical analysis to the direct evaluation of the Nusselt number.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the financial support provided by CNPq, CAPES, and FAPEMIG. The authors also acknowledge the Federal University of Itajubá (UNIFEI) for providing the infrastructure and facilities necessary for the development of this research.

REFERENCES

Khudheyer, A. F., and Hasan, Z. H., 2015, “Effect of the Fins Configuration on Natural Convection Heat Transfer Experimentally and Numerically,” International Journal of Energy and Environment, Vol. 6, No. 6, pp. 607–628.

Kim, T. H., Do, K. H., and Kim, D. K., 2011, “Closed Form Correlations for Thermal Optimization of Plate-Fin Heat Sinks under Natural Convection,” International Journal of Heat and Mass Transfer, Vol. 54, Nos. 5–6, pp. 1210–1216.

Kim, T. H., Kim, D. K., and Do, K. H., 2013, “Correlation for the Fin Nusselt Number of Natural Convective Heat Sinks with Vertically Oriented Plate-Fins,” Heat and Mass Transfer, Vol. 49, pp. 413–425.

Leung, C. W., Probert, S. D., and Shilston, M. J., 1985, “Heat Exchanger Design: Optimal Uniform Separation between Rectangular Fins Protruding from a Vertical Rectangular Base,” Applied Energy, Vol. 19, No. 4, pp. 287–299.

Ong, K. S., Tan, C. F., Lai, K. C., and Tan, K. H., 2017, “Heat Spreading and Heat Transfer Coefficient with Fin Heat Sink,” Applied Thermal Engineering, Vol. 112, pp. 1638–1647.

Shen, Q., Sun, D., Xu, Y., Jin, T., and Zhao, X., 2014, “Orientation Effects on Natural Convection Heat Dissipation of Rectangular Fin Heat Sinks Mounted on LEDs,” International Journal of Heat and Mass Transfer, Vol. 75, pp. 462–469.

Tari, I., and Mehrtash, M., 2013, “Natural Convection Heat Transfer from Inclined Plate-Fin Heat Sinks,” International Journal of Heat and Mass Transfer, Vol. 56, Nos. 1–2, pp. 574–593.