

Comparison Between Numerical Study and Experimental Work on Heat Transfer from Heat Sink Under Transient Conditions

Ehsan F. Abbas*, Hussein H. Mohammed Ali, Nawzad J. Mahmood

Department of Power Mechanics Techniques Engineering- Technical Engineering College (Kirkuk) Northern Technical University, Iraq.

*Corresponding author email: ehsanfadhil@ntu.edu.iq

ABSTRACT

Fins are widely used in most of the thermal equipment's in the field of enhancing heat dissipation by extending a surface area. Micro-channel sink heat is an important part used in the cooling process in various devices, especially in electronic devices, to cool the processor part, power supply...etc. It's available in a different shape and made of materials with high thermal conductivity, such as copper and aluminum. The present study examines the numerical solution to a small heat sink involving practical experiments at constant working conditions and air volumetric flow rates ranging from 20 to 100 cc/s in five increments of 20 cc/s at constant heat flux of 9000 W/m². The COMSOL 5.2 software was used to simulate a heat sink model that uses solid-mode transient heat transfer. The process experiments were carried out under similar working conditions, which was to start recording the experimental data when the temperature of the heat sink base reached 40°C, and to continue for a period of 1200 seconds. Both the numerical simulation results and the practical work were compared and a good match was found between them. A higher error rate was obtained at a volumetric flow rate of 20 cc/s, which is not more than 1.2%.

KEYWORDS

Transient heat simulation, fin analytic solution, COMSOL software, transient temperature distribution.

INTRODUCTION

Fins are used in many engineering applications, such as in the field of thermal engineering. Fin's main objective is to improve heat dissipation from the surface to the surroundings. The designs of fins in engineering equipment include different geometries, such as longitudinal, radial, and pin fins. All these fin types are found in both constant and variable cross-sectional areas. A heat sink straight fin is widely used in electronic devices such as computers, cool processors, or power supplies. Many theoretical and experimental studies have been carried out to verify the performance of heat sinks, and some can be summarized as follows. Fahimina et al. [1] have experimentally and numerically studied improving the free convection heat transfer based on the optimum fin spacing in the heat sink used in a computer. They found a non-dimensional parameters correlation between the fin arrays' heat dissipation compared to the base plate without a fin. Lee et al. [2] experimentally studied forced convection heat transfer on the heat sink with partial heating including a measure of heating length, heating position, flow rate on the temperature distribution, and thermal performance. Raut and Kothavale [3] studied the thermal performance of a micro-finned heat sink under natural convection.

The current study examined heat transfers for various fin parameters such as fin thickness, fin spacing, and fin height. They found that the maximum thermal performance occurred when the fin was with the highest spacing and minimum thickness. Al-Sallami et al. [4] presented a numerical solution to evaluate the heat transfer enhancement of the heat sink using a conjugate heat transfer model. This study showed that a strip heatsink type staggered arrangement gives the highest enhancing heat dissipation and contributes to reducing power consumption. Sahu et al. [5] used different mathematical methods to solve the multi-dimensional fin heat transfer. From their results, the finite difference approach is the best method for this complex problem. Kandasamy and Prabhu [6] simulated a different type of pin fin based on bio-inspired airflow using COMSOL multiphasic software. Several parameters were examined in the present study, such as porosity, aspect ratio, and heat flux under Reynolds number 2300. The result showed that lower porosity gives higher heat dissipation from the heat

sink to the surroundings. Nakharin et al. [7] studied the effect of the jet plate spacing to jet diameter ratio on the heat dissipation from the micro-channel heat sink using nanofluid type TiO₂.

Experiments were conducted on an H/D basis ranging from 0.8 to 4, a nanofluid concentration from 0.005 to 0.015 by volume, and mass flow rates from 8 to 12 g/s. They found that the jet plate ratio to a jet diameter significantly influences temperature flow behavior and increased heat transfer. Naphon and Nakharin [8] presented a numerical solution for analyzing nanofluids heat dissipation from the mini-channel heat sink in a turbulent to phase flow. Experiments were performed with the nanofluid Reynolds number ranging from 80 to 200. The result of nanofluid compared with deionized water showed that the nanofluid has more efficiency in the cooling method and the suspending nanofluids have a significant effect on enhancing heat dissipation. Duan et al. (2020) [9] analyzed flow characteristics and pressure drop in a heat sink with elliptic bottom profiles using numerical simulation. The simulation was performed with different width entries. The result showed that the pressure drop decreases at the same airflow rate as the inlet width increases. They also get a pressure drop for collided plates that do not have an elliptical bottom much higher than an elliptical base. Teertstra et al. [10] presented an analytical model to predict an average heat transfer rate in a forced convection.

The present study includes selecting a heat sink to be used in electronic applications. A composite solution has been used in a simulation-based on the limiting boundary conditions, such as fully developed and developed flow between two isothermal parallel plates. The result proved that the analytical model applicable to Reynolds No. $0.1 < Re_d^* < 100$. Kim and Mudawar [11] used an analytical simulation to evaluate the heat dissipation of different microchannel heat sink geometries in an electronic cooling application. Fin geometries are rectangular, inverse trapezoidal, triangular, trapezoidal, and diamond-shaped cross-sections. The comparison showed that the analytical models give an accurate prediction of the design and thermal resistance of microchannel heat sinks in an electronic cooling application. Sharath et al. [12] presented a numerical simulation to calculate heat transfer and optimization of heat sinks of different orientations and geometries. The aim of the present study was to increase heat transfer and reduce pressure drop through studying circular, elliptical and square cross-section fins of inline and staggered type. Korzen and Taler [13] presented a mathematical model to simulate the heat transfer coefficient from the plate heat exchanger under the transient case, and its result has been compared with similar experimental work.

They get a good agreement between the computation results and the experimental result. Pati et al. [14] investigated the effect of geometry pin fins for an inline and staggered arrangement on the microprocessor's thermal performance. Experiments have been performed by varying Reynolds number, interspace ratio and arrangement of fins on the heat dissipation. Comparative results showed a significant increase in Nusselt number observed with increasing Re number for each fin arrangement. Saravanan et al [15] numerically studied the effect of two different geometries of pin fin, microchannel pin fin heat sink and micro pin fin heat sink and examined the properties of heat transfer and pressure drop. Fins of different shapes namely with square and circular cross-sections and with inline and staggered arrangements are used. Experiments were conducted at a constant heat flux of 10 W/m^2 , using water as a coolant fluid, for the Reynolds numbers ranging between 100 and 900. Both fins' heat transfer rates were compared; the result showed that the pin fin heat sink gives better thermal performance than the microchannel pin fin heat sink.

The results also showed that the pin heat sink has higher efficiency compared to the microchannel pin heat sink and is efficient preferable when the heat dissipation to the penalty of pressure drop. Shah et al. [16] studied transient analytical solutions to heat transfer in fins made from different materials using the finite element method (FEM). They studied the heat transfer rate parameters, the time to reach a steady-state, the temperature distribution inside the fin at different times, and the steady-state temperature to obtain the best fin material. Mahara and Singh [17] simulated transient heat transfer from the aluminum alloy cylindrical fin, and it was compared with different fin material like aluminum, brass, and copper, using Ansys APDL software. The present study compares the numerical simulation of a straight heat sink with a fixed cross-section using COMSOL Multiphysics 5.2 with experimental work. The numerical simulation was performed in case of forced convection at a constant heat flux of 600 W/m^2 and a volumetric flow rate of 20 to 100 cc/s.

MATERIAL AND METHOD

This study includes two methods, the first is the design of a numerical model by COMSOL software, and its results have been experimentally examined as a second method.

Numerical models

Used a COMSOL Multiphysics 5.2 to simulate numerical heat dissipating from the straight fin heat sink based on the actual boundary conditions obtained from experiments conducted on the standard test rig. A numerical model was prepared through a sequential application of the software steps as in the following procedures:

- The program stated- Modal wizard- 3D is Chosen-Heat transfer in solid -Time dependent.
- Building the geometry of the heat sink according to its actual dimensions shown in Table 1 as shown in figure 1.
- Aluminum selection to define the material of the heat sink parts.
- The boundary conditions shown in Table 2 were used to simulate the heat dissipation from the heat sink model.
- Meshing have been established
- The simulation starts with an increment of time 1 seconds.

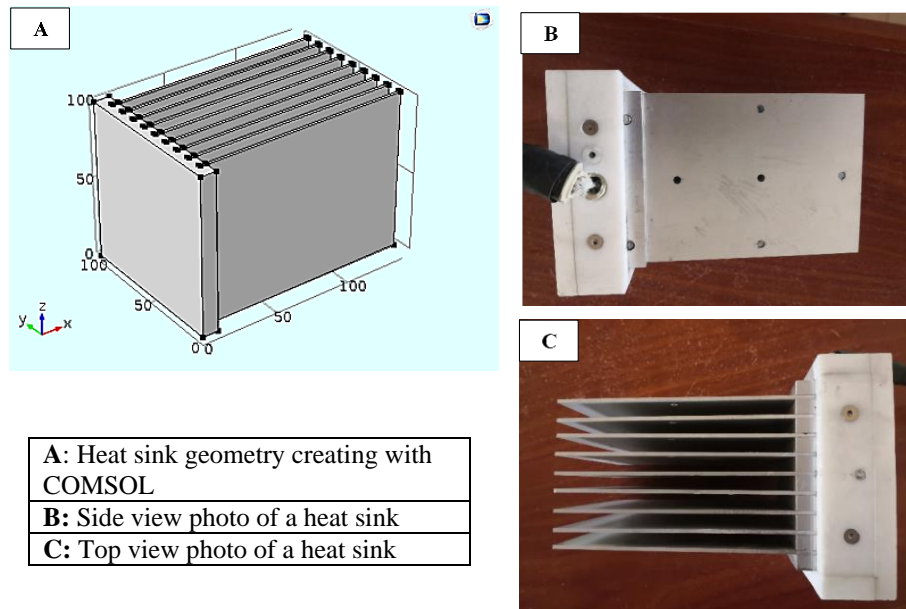


Figure 1. Sketch diagram and photo of a heat sink

Table 1. Technical specifications of the heat sink

Part name	Dimension	Material
Base	10cm × 10cm × 1cm	Aluminum
Fins	135 cm × 10 cm × 2 mm; No. of fins 9	Aluminum
Electric element	8 cm length and 8 mm diameter, 140 W	Stainless steel

Experimental Setup

Practical tests were performed by a computerized thermal apparatus using the heat sink shown in figure 2 with technical specifications in Table 1. Five tests were performed based on the working boundary conditions shown in Table 2 with constant heat flow at a volumetric airflow rate of 20 to 100 cc/s with a flow step of 20 cc/s. The test apparatus automatically saves the experiment data and controls its operation through its software. In the present work, the 60th second step has chosen to preserve data throughout the testing time. Uniform procedures have been followed to execute tests. First, the heat sink was heated until it reached 40°C for a half-hour at a heat flux of 9000 W/m² without a fan operating. After this period, the test starts for 1- 200 seconds. Use these tests data to examine the numerical results obtained from COMSOL.



Figure 2. Photo of the user test device

Table 2. Working boundary conditions

Parameters	Unit
Heat flux	600 W/m ²
Temperature of the air follows	26°C ±1°C
Range of volumetric air flowrate	20 to 100 cm ³ /s
Testing time	1200 s

RESULTS AND DISCUSSION

In the present study, COMSOL 5.2 is used to simulate the transient temperature distribution in a uniform cross-section in a small heat sink and examine its results in practical experiment. Numerical simulations were performed for five volumetric airflow rates ranging from 20 cc/s to 100 cc / s and their results are illustrated in figures 3 to 9, which express temperature behavior within the fins. For five different time increments during the specified test period of 60, 300, 600, 900 and 1200 seconds. Previous figures indicated that the airflow influenced the temperature distribution behavior inside the fin due to increased heat dispersed from the heat sink. It has been observed from these tests that the level of temperature distributions decreases when the volumetric airflow rate increases, and most of the heat is dissipated from the unfinned and finned area near it, this is evident from the large difference in temperature distribution along the fin with respect to the increase in the rate of airflow.

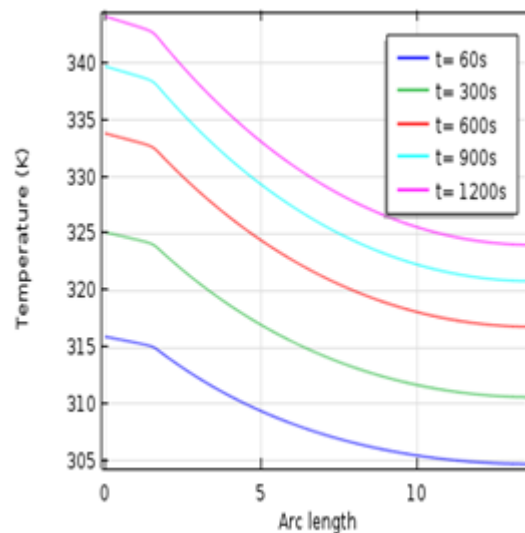


Figure 3. Temperature distribution behavior along the fin at an airflow rate of 20 cc/s

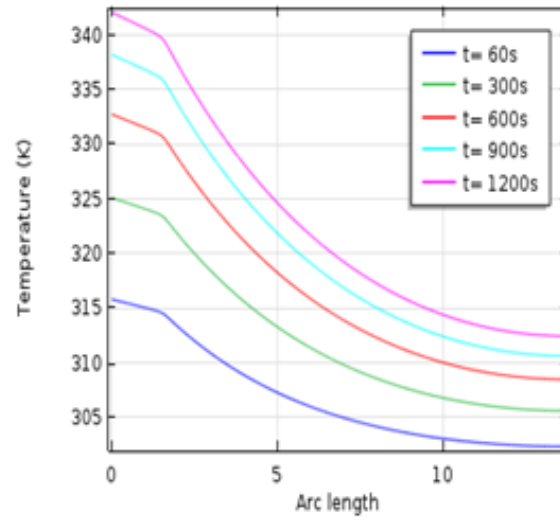


Figure 4. Temperature distribution behavior along the fin at an airflow rate of 40cc/s

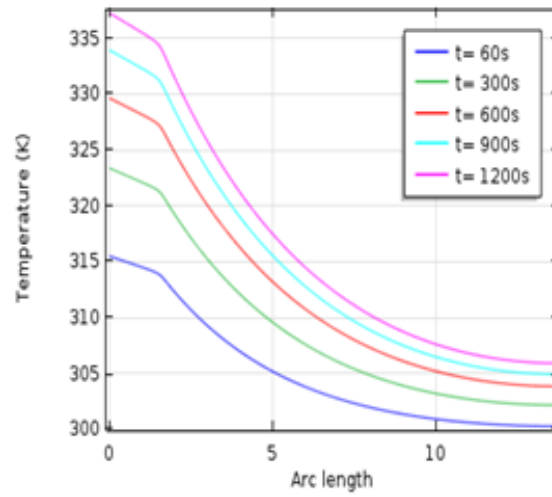


Figure 5. Temperature distribution behavior along the fin at an airflow rate 60 cc/s

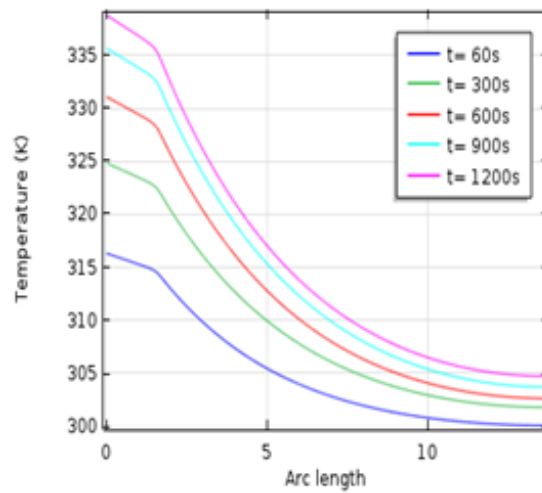


Figure 6. Temperature distribution behavior along the fin at an airflow rate of 80 cc/s

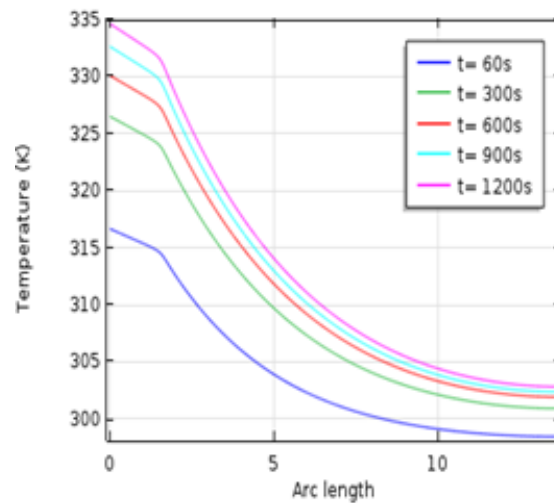


Figure 7. Temperature distribution behavior along the fin at an airflow rate of 100 cc/s

The COMSOL software results were examined along with the practical test results, which were performed based on the working boundary conditions shown in Table 2. The comparison results are shown in figures 8 to 12. In all of these figures, there is a good match between the results of both the COMSOL tests and the practical work, it has been noted that the percentage of error between the two is very small. The results indicate that the highest error ratio obtained at a volumetric flow rate of 20 cc/s, ranges from 0.2% to 1.2%, while in the other airflow rates, the percentage of error does not exceed 0.8%. The error between the two results comes from using an average heat transfer coefficient in the COMSOL simulations. In fact, this value changes across the fin.

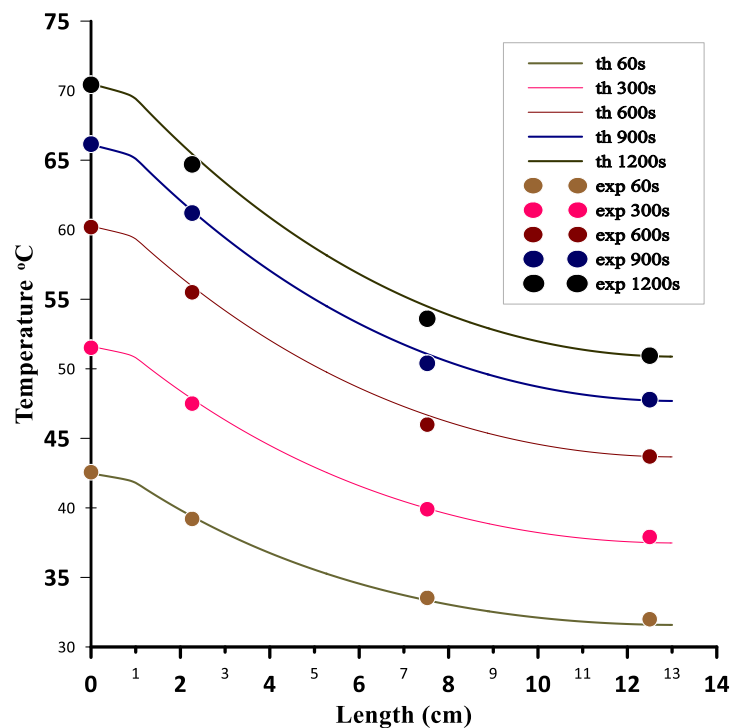


Figure 8. Theoretical and experimental Temperature distribution along the fin at airflow rate of 20 cc/s

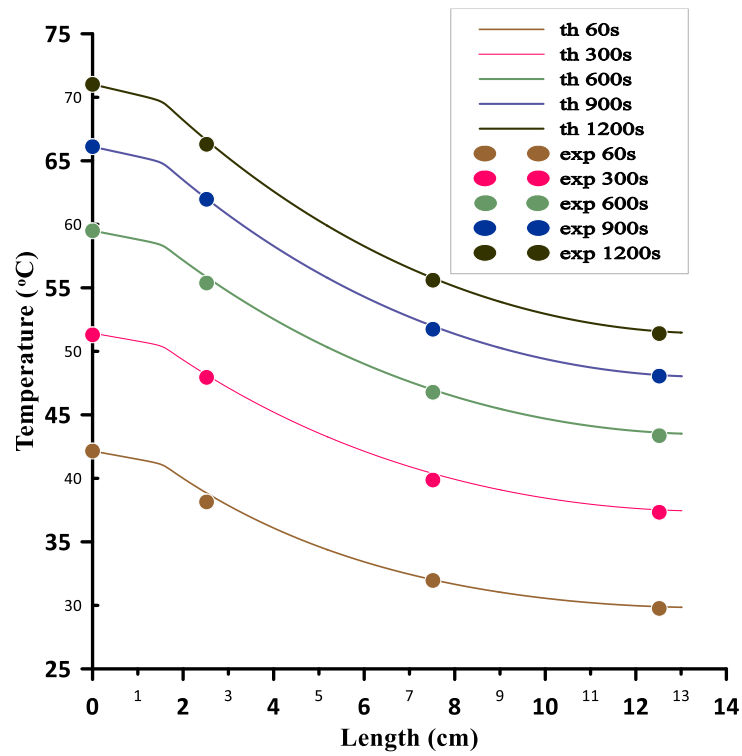


Figure 9. Theoretical and experimental Temperature distribution along the fin at airflow rate of 40 cc/s

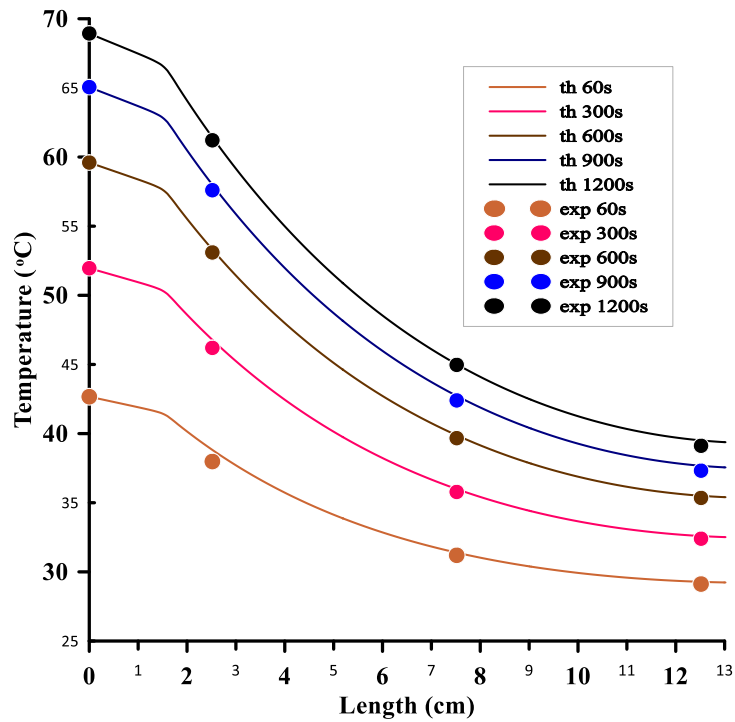


Figure 10. Theoretical and experimental Temperature distribution along the fin at airflow rate of 60 cc/s

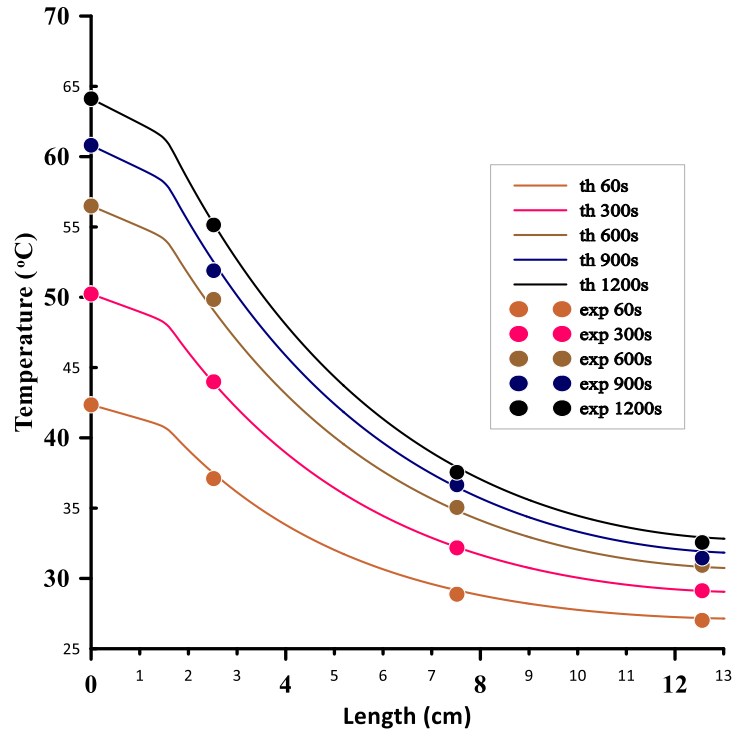


Figure 11. Theoretical and experimental Temperature distribution along the fin at airflow rate of 80 cc/s

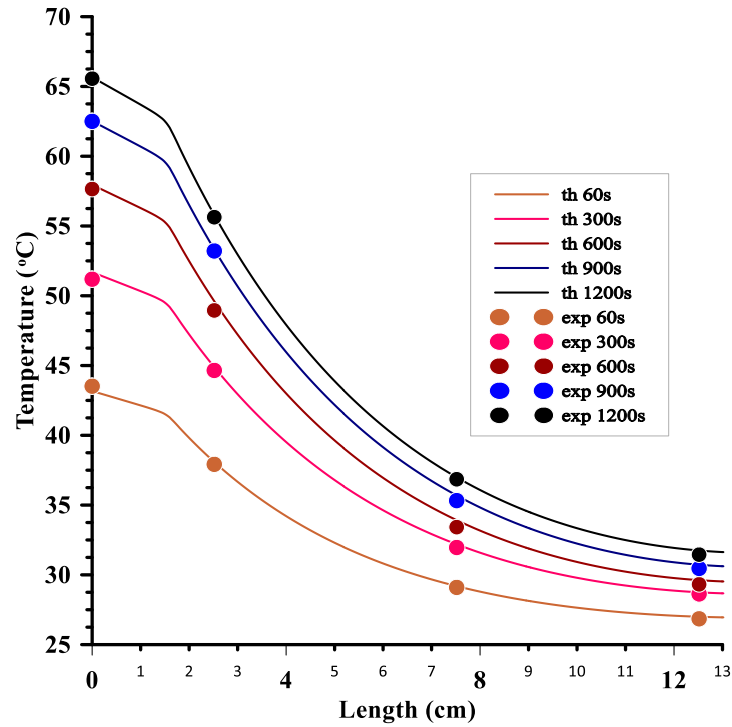


Figure 12. Theoretical and experimental Temperature distribution along the fin at airflow rate of 100 cc/s

CONCLUSION

Transient temperature distribution along constant rectangular fin has been investigated using COMSOL software and examined experimentally through practical tests on the heat sink using a convection heat transfer apparatus. Comparative results have shown that there is a good agreement between the practical results and the numerical solution, where the maximum percentage of error does not exceed 1.2% but when the volumetric airflow rate increases more than 20 cc/s to reach 100 cc/s, the error ratio decreases to 0.8%.

RECOMMENDATIONS FOR FUTURE WORK

The weaknesses and limitations of experimental time and mesh size developed in this study have indicated that some errors between experimental and numerical study, we recommended for future work, re-study using a longer time and reducing the size of the network, in addition to using more than one heat exchanger than was used in the current study.

REFERENCES

- [1] M. Fahiminia, "Experimental investigation of natural convection heat transfer of the fin arrangement on a computer heat sink," *Sci. Res. Essays*, vol. 7, no. 36, pp. 3162–3171, 2012. <https://doi.org/10.5897/SRE11.2167>
- [2] J. J. Lee, H. J. Kim, and D. K. Kim, "Experimental study on forced convection heat transfer from plate-fin heat sinks with partial heating," *Processes*, vol. 7, no. 10, pp. 1–18, 2019. <https://doi.org/10.3390/pr7100772>
- [3] S. V Raut and B. S. Kothavale, "Study on Thermal Performance of Micro Fin Heat Sink under Natural Convection – A Review," *Int. J. Curr. Eng. Technol.*, vol. 7, no. 7, pp. 257–261, 2017. <http://dx.doi.org/10.22055/JACM.2019.14852>
- [4] W. Al-Sallami, A. Al-Damook, and H. M. Thompson, "A numerical investigation of thermal airflows over strip fin heat sinks," *Int. Commun. Heat Mass Transf.*, vol. 75, pp. 183–191, Jul. 2016. <https://doi.org/10.1016/j.icheatmasstransfer.2016.03.014>
- [5] M. Sahu, G. Chand Sahu, M. Sao, and A. K. Jain, "Analysis of Heat Transfer From Fins Using Finite Difference Method", *International Journal of Advance Research, Ideas and Innovation in Technology, KITE/NCISRDC/IJARIIT/2018/Mech/105, IJARIIT (ISSN: 2454-132X)*, pp 261-269, 2018.
- [6] K. Kandassamy and B. Prabu, "Numerical investigation of bio-inspired pin fin heat sink models for square shaped electronic circuits," *J. Appl. Sci. Eng.*, vol. 22, no. 1, pp. 119–133, 2019. DOI: 10.6180/jase.201903_22(1).0012
- [7] L. Nakharintr, P. Naphon, and S. Wiriyaart, "Effect of jet-plate spacing to jet diameter ratios on nanofluids heat transfer in a mini-channel heat sink," *Int. J. Heat Mass Transf.*, vol. 116, pp. 352–361, 2018. DOI: 10.1016/j.ijheatmasstransfer.2017.09.037
- [8] P. Naphon and L. Nakharintr, "Turbulent two phase approach model for the nanofluids heat transfer analysis flowing through the minichannel heat sinks," *Int. J. Heat Mass Transf.*, vol. 82, pp. 388–395, 2015. DOI: 10.1016/j.ijheatmasstransfer.2014.11.024
- [9] Z. Duan, X. Lv, H. Ma, L. Su, and M. Zhang, "Analysis of flow characteristics and pressure drop for an impinging plate fin heat sink with elliptic bottom profiles," *Appl. Sci.*, Vol.10, no. 1, 2020. DOI: 10.3390/app10010225
- [10] P. Teertstra, M. M. Yovanovich, and J. R. Culham, "Analytical forced convection modeling of plate fin heat sinks," *J. Electron. Manuf.*, vol. 10, no. 4, pp. 253–261, 2000. DOI: 10.1142/S0960313100000320
- [11] S.-M. Kim and I. Mudawar, "Analytical heat diffusion models for different micro-channel heat sink cross-sectional geometries," *Int. J. Heat Mass Transf.*, vol. 53, no. 19–20, pp. 4002–4016, Sep. 2010. DOI: 10.1142/S0960313100000320
- [12] D. Sharath, Sathyanarayana, and H. S. Puneeth, "Heat Transfer Numerical Simulation and Optimization of a Heat Sinks," in *IOP Conference Series: Materials Science and Engineering*, (2018) 376(1). DOI: 10.1088/1757-899X/376/1/012005
- [13] A. Korzeń and D. Taler, "Modelowanie matematyczne pracy wymiennika ciepła z rur ożebrowanych dla zmiennych w czasie strumienia masy płynu i prędkości powietrza," *Czas. Tech.*, vol. 3/2017, pp. 183–196, 2017. DOI: 10.4467/2353737xct.17.041.6352
- [14] A. Gupta, M. Kumar, and A. K. Patil, "Enhanced heat transfer in plate fin heat sink with dimples and protrusions," *Heat Mass Transf. und Stoffuebertragung*, Vol. 55, no. 8, pp. 2247–2260, 2019. DOI: 10.1007/s00231-019-02561-w

- [15] V. Saravanan and C. K. Umesh, "Numerical comparison for thermo-hydraulic performance of pin fin heat sink with micro channel pin fin heat sink," *Sadhana - Acad. Proc. Eng. Sci.*, vol. 43, no. 7, pp. 1–15, 2018. DOI: 10.1007/s12046-018-0875-1
- [16] M. P. Shah, K. S. Mehra, S. Gautam, and P. Negi, "Transient and Steady State Analysis of Fin Using FEM for Different Material," *Int. J. Res. Appl. Sci. Eng. Technol.*, vol. 2, no. Vi, pp. 36–40, 2014.
- [17] N. K. Mahara and V. Singh, "Transient state analysis of cylindrical fin using simulation method and compare different fin material," *Glob. J. Eng. Sci. Res.*, vol. 4, no. 1, pp. 80–89, 2017. DOI: 10.5281/zenodo.263964