Influence of inlet velocity and heat flux on the thermal characteristic of various heat sink designs using CFD analysis

C.Y. Khor¹, M.U. Rosli¹, M.A.M. Nawi¹, W.C. Kee¹, Dadan Ramdan²

¹Simulation and Modelling Research Group (SiMMREG), Faculty of Mechanical Engineering Technology, Universiti Malaysia Perlis (UniMAP), Perlis, Malaysia.
²Fakultas Teknik, Universitas Medan Area, Jl. Kolam No. 1, Medan Estate, Medan 20371, Indonesia.

Email: cykhor@unimap.edu.my

Abstract. A heat sink is an important passive heat exchanger in the thermal management of electronic devices or systems. The study aims to analyze the influence of the inlet velocity and heat flux on the thermal characteristics (i.e., static temperature and heat transfer coefficient) of three heat sink designs. Computational fluid dynamics (CFD) analysis is carried out for the electronic cooling process. Three heat sink designs, namely, circular pin fin, plate fin, and rectangular fin are considered in this study. The influence of the inlet velocity and heat flux on the temperature distribution and heat transfer coefficient was analyzed. The results revealed that the use of circular pin resulted the highest temperature drop when the inlet velocity increased. The highest heat transfer coefficient was observed on the circular pin fin design during the cooling process. The heat flux demonstrated a linear correlation with the temperature for all heat sink designs. The effectiveness of the heat transfer was attributed by the heat sink design and inlet velocity of the airflow. Thus, the current results indicated the thermal characteristics of the heat sink are crucially influenced by the design and velocity of airflow in the cooling process.

1. Introduction

The heat sink plays an important role in the thermal management to dissipate the heat from an electronic device or system. The heat dissipates to the surrounding and the device maintains at the operating temperature. The efficiency of heat dissipation is closely related to fin design and the use of external mean such as cooling fans and cooling channels. The heat sink with various fin designs was considered in the previous studies, such as the design of plate fin shape [1, 2], square pin fin shape [3], in-line and staggered arrangements fin [4]. Besides, the staggered arrangement [5] and chimney based design [6] were also considered in the radial heat sink design. Various heat sink designs could have different thermal characteristics. The preliminary input of the thermal characteristic is important for the engineer and researcher and it could be collected via simulation and modeling analysis [7, 8].

With the rapid development of the computational analysis tools, the real condition of the process could be mimics by the simulation models in various case studies such as wave soldering [9] and reflow soldering process [10, 11], heat performance of the infrared oven [12], micro-heat sink [13] and
encapsulation process [14, 15]. The simulation model can be solved by different discretization methods, which are finite volume and finite element methods [16, 17]. Sometimes, integrated of both methods were also used to solve the fluid-structure interactions problems [18]. In the study of heat sink area, a simulation technique is applied to visualize the temperature distribution and flow characteristics during the cooling process. Reynolds number, fin width and height [19] were significant factors that affect the thermal performance. The increase in Reynolds number reduces the thermal resistance of the heat sink. The optimum design of fin width is important to allow air flowing through the heat sink, the narrow flow passages will reduce heat transfer. Moreover, the cross-cut length was most significant influential factors among cross-cut design parameters [20]. Besides, the effects of different fins such as extrusion, plain, and cell on thermal performance were studied by Yeh [21]. Fin design and flow-by-pass was crucially affected the heat transfer coefficient and thermal resistance.

The current study is the extension from our previous work [22]. A similar heat sink design (i.e., plate, circular pin and rectangular) was considered in the computational fluid dynamics (CFD) analysis. The heat sink models were meshed and simulated in CFD software to mimic the cooling process. The effect of various inlet velocities and heat fluxes toward the temperature and heat transfer coefficient was studied.

2. Methodology

A similar dimensions of three heat sinks [22] was used in this study (Figure 1). The detailed dimensions of the heat sink design were presented in our previous work [22]. Three heat sinks were constructed in three-dimensional (3D) model, as illustrated in Figure 1. The rectangular domain was created to allow the air flowing through the heat sink. The surfaces of the domain were defined by the inlet, outlet, and no-slip wall boundaries. The heat flux was defined at the heat sink base. In the simulation, the air enters the domain from inlet boundary and leaving the domain at the outlet boundary. In this study, the inlet velocity (i.e., 1 – 9 m/s) and heat flux (100 – 900 W/m²) was considered in the parametric analysis.

The governing equations [7-9] (i.e., continuity, momentum and energy equations) were used to model and describe the motion of airflow in the CFD simulation. The airflow was assumed as a steady flow, incompressible flow and in a laminar condition. SIMPLE scheme and 2nd order upwind discretization were applied in the solution discretization method [9]. The solutions converged when the residuals plot reach 1E-6 [22]. The maximum temperature, heat transfer coefficient and pressure drop were analyzed.

![FIGURE 1. Three heat sink designs: (a) plate, (b) circular pin and (c) rectangular [22].](image-url)
3. Results and Discussion

3.1. Influence of inlet velocity

The simulation of the cooling process for three heat sink models were conducted. A constant heat flux (500W/m$^2$) was considered for various inlet velocity cases. In the simulation, the velocities of air flow are 1, 3, 5, 7, and 9 m/s, where the air flows in the longitudinal direction of fluid domain. The results revealed the decrease in maximum temperature of the heat sink when the inlet velocity rise. The maximum temperature gradually decreases in the polynomial behaviour for all heat sinks (Figure 2). Circular pin fin heat sink shows a dramatic decline of the temperature around 70 K in total. Circular pin fin encounters the highest temperature at 1 m/s of inlet velocity. However, it drops until the least temperature when 9 m/s of air velocity were applied. It shows that the circular pin fin heat sink has an obvious temperature drop while applied with higher inlet velocity. This situation was attributed to the large airflow passages compared with rectangular and plate fin heat sink. Larger airflow passages allow the air flowing easily and carry the heat away from the heat sink at high inlet velocity. Hence, the circular pin fin heat sink is recommended if inlet velocity is greater due to the better thermal cooling effect.

Figure 3 shows the coefficient of heat transfer for three heat sinks. The circular pin heat sink showed the increasing of heat transfer coefficient (HTC) when increase in inlet velocity. However, lowest HTC was observed for the rectangular fin heat sink after 4 m/s, which may be due to small spacing between the rectangular fins. The narrow airflow passages have restricted the air passing through the heat sink. As shown in Figure 3, with increasing of inlet velocity, the surface heat transfer coefficient is enhanced, which increases the convection heat transfer of the fluid and ultimately the temperature is reduced.

The total pressure drop, which includes the loss of a combination of friction, inlet and exit which is presented in Figure 4. The pressure drop of the airflow increases in all heat sink designs in the polynomial behaviour with an increase in inlet velocity. As expected, the circular pin fin heat sink has the highest pressure drop among all three heat sinks because of the small free flow cross-section area and the circular cylindrical shape of fins caused the pressure difference changes the greatest before and after the fluid striking on the fins. Hence, improving the splitter’s distance will increase the pressure drop, but weakening of the vortex behind the pin, which minimizes the total pressure drop.

![Figure 2. Maximum temperature of three heat sinks.](image)
3.2 Influence of heat flux

In the CFD simulation, the effect of heat flux was investigated at a constant inlet velocity when the heat flux varied from 500 – 900 W/m². Five heat flux values were considered, which are 500W/m², 600W/m², 700 W/m², 800 W/m² and 900 W/m² to investigate those effects on the temperature of the heat sink. The simulation results revealed that the temperature of all heat sinks increases linearly as the increasing of heat flux (Figure 5). The highest temperature was obtained at the circular pin heat sink around the outlet in the case of highest heat flux (900W/m²). As the heat flux increases, the outlet temperature continues rising because the airflow (fluid medium) gets heated up increasingly due to convective heat transfer, and it makes the airflow unable to carry more heat away from outlet. Inadequate of the cooling may lead to overheat of the heat sink and cause electronic devices or systems exceeding the operating temperature. To prevent the unintended overheating issue, design and

![Figure 3. Surface heat transfer coefficient of three heat sinks.](image1)

![Figure 4. Pressure drop of three heat sinks.](image2)
process optimization [23, 24] is useful to analyse the most significant factors and interaction between design and process parameters of the heat sink. Thus, the basic understanding of the heat sink design is important for the engineer.

Figure 5. Static temperature of three heat sinks.

4. Conclusion

The influence of the inlet velocity and heat flux on the temperature and heat transfer coefficient has been studied by using the CFD simulation. The current results demonstrated that inlet velocity and heat flux crucially affect the temperature and heat transfer performance on various designs of heat sink. The circular pin heat sink showed a high temperature drop and heat transfer coefficient when rise in inlet velocity. Besides, the increase in heat flux significantly rise the heat sink temperature. The wide airflow passages allow air flow easily and it enhanced the effectiveness of heat transfer. The comparison study revealed that the circular pin fin heat sink yield better performance compared to plate fin and rectangular fin. Therefore, the simulation results are expected to provide preliminary input and understanding to the thermal management engineer before designing the heat sink for electronic devices or systems. The current study will be extended on the optimization of the heat sink design by considering various design and process parameters.

References

[1] Jeon D and Byon C 2017 *International Journal of Heat and Mass Transfer* **113** 1086-92.
[2] Hoi S M, Teh A L, Ooi E H, Chew I M, Foo J J 2019 *International Journal of Thermal Sciences* **142** 392-406.
[3] Sakanova A and Tseng K J 2018 *Applied Thermal Engineering* **136** 364-74.
[4] Kewalramani G V, Hedau G, Saha S K and Agrawal A 2019 *International Journal of Heat and Mass Transfer* **138** 796-808.
[5] Maji A, Bhanja D and Patowari P K 2017 *Applied Thermal Engineering* **125** 596-616.
[6] Schmid G, Valladares-Rendón L G, Yang T H and Chen S L 2017 *Applied Thermal Engineering* **125** 575-83.
[7] Abdul Aziz M S, Abdullah M Z, Khor C Y, Jalar A and Che Ani F 2014 *International Journal of Heat and Mass Transfer* **72** 400-410.
[8] Khor C Y, Abdullah M K, Abdullah M Z, Abdul Mujeebu M, Ramdan D, Majid M F M A and Ariff Z M 2010 *Heat and Mass Transfer/Waerme- und Stoffubertagung* **46** 1315-1325
[9] Abdul Aziz M S, Abdullah M Z, Khor C Y, Fairuz Z M, Iqbal A M, Mazlan M and Rasat M S M 2014 Advances in Mechanical Engineering 2014 275735
[10] Lau C-S, Abdullah M Z and Khor C Y 2013 Microelectronics International 30(3) 151-168
[11] Lau C S, Khor CY, Soares D, Teixeira J C and Abdullah M Z 2016 Soldering and Surface Mount Technology 28(2) 41-62
[12] Najib A M, Abdullah M Z, Khor C Y and Saad A A 2015 International Journal of Heat and Mass Transfer 87 49-58
[13] Alzahrani M, Baig H, Shanks K and Mallick T 2020 Applied Thermal Engineering 176 115315.
[14] Khor C Y and Abdullah M Z 2012 Simulation Modelling Practice and Theory 29 109-122
[15] Ong E E, Abdullah M Z, Khor C Y, Loh W K, Ooi C K, Chan R 2012 International communications in heat and mass transfer 39(10) 1616-1623
[16] Khor C Y and Abdullah M Z 2013 Microelectronics Reliability 53(2) 334-347
[17] Ramdan D, Abdullah M Z, Khor C Y, Leong W C, Loh W K, Ooi C K and Ooi R C 2012 IEEE Transactions on Components, Packaging and Manufacturing Technology 2(11) 1786-1795
[18] Ong E E S, Abdullah M Z, Khor C Y, Loh W K, Ooi C K and Chan R 2014 Microelectronic Engineering 113 40-49
[19] Li H and Chao S 2009 International Journal of Heat and Mass Transfer 52(13-14) 2949-2955
[20] Kim T and Kim S 2009 International Journal of Heat and Mass Transfer 52(23-24) 5358-5370
[21] Yeh L-T 2012 13th InterSociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems, San Diego, CA, 2012 446-449
[22] Khor C Y, Nawi M A, Ishak M I, Kee W C, Rosli M U, Jamalludin M R and Termizi S A. 2020 IOP Conference Series: Materials Science and Engineering 932(1) 012106
[23] Leong W C, Abdullah M Z and Khor C Y 2013 Microelectronics Reliability 53(12) 1996-2004.
[24] Aziz M A, Abdullah M Z, Khor C Y and Azid I A 2015 Simulation Modelling Practice and Theory 57 45-57.