International Journal of Innovative Research in Technology and Science

ISSN: 2321-1156 www.ijirts.org Volume 12 Issue 2, March 2024

Computational Fluid Dynamics Study of Forced Convection Cooling for Electronic Chips

Pankaj Kumar Bangre, Vineet Kumar Dwivedi

Faculty of Engineering and Technology, SAM Global University, Bhopal, India

Selection and peer review of this article are under the responsibility of the scientific committee of the International Conference on Current Trends in Engineering, Science, and Management (ICCSTEM-2024) at SAM Global University, Bhopal.

Abstract- Over the past decade, significant research has been dedicated to comprehending the fluid dynamics and heat transfer behaviours within silicon-based microchannel heat sinks, especially in the realm of electronic cooling applications. These microchannel heat sinks, characterised by their non-circular channels and silicon composition, offer an enticing blend of material compatibility, high surface area-to-volume ratios, and efficient heat transfer potential, all attainable through cost-effective fabrication processes. This renders them highly attractive for a diverse range of commercial applications. This study uses the FLUENT commercial CFD software to concentrate on cooling electronic chips, utilising forced water convection within single microchannel heat sinks made of silicon. The computational domain is discretised utilising non-uniform grids on the flow face and uniform grids along the flow direction, with grid generation facilitated by Gambit software and the Cooper method. Pressure, velocity, and temperature distributions at the inlet and outlet are scrutinised, along with variations along the flow direction. The results are validated against existing data for silicon substrates.

Furthermore, Nusselt number variations and convection heat transfer coefficients are examined for different flow rates. The simulation uses the Semi-Implicit Method for Pressure-linked Equations (SIMPLE) with a second-order upwind scheme for fully-developed laminar Flow, solving continuity, momentum, and energy equations separately. Post-processing of results is carried out using Excel. The study considers multiple pressure drops to optimise power consumption, aiming to minimise temperature rise in the sink, which is crucial for preventing water temperature from reaching the boiling point. Overall, the simulation results align well with available data, presenting detailed profiles of velocity, temperature, and pressure differences along the channel.

Keywords: Micro Channel Heat Sinks, Forced Convention Cooling, Silicon-Based Cooling, CFD, Heat Transfer, Electronic Chip Cooling

1. INTRODUCTION

In recent years, there has been a growing demand for efficient cooling solutions in electronic devices, particularly as the miniaturisation and integration of electronic components continue to advance. Managing the associated heat generation becomes a critical engineering challenge as electronic devices become increasingly compact and powerful. Heat dissipation is crucial to maintaining the reliability and performance of electronic systems, as excessive temperatures can lead to thermal stress, component degradation, and even system failure. Among various cooling techniques, forced convection cooling utilising microchannel heat sinks has emerged as a promising solution due to its effectiveness in dissipating heat from electronic chips. Microchannel heat sinks offer several advantages over traditional cooling methods. These heat sinks typically feature non-circular channels and are often fabricated from materials such as silicon, which possess favourable thermal properties and compatibility with electronic components. The design of microchannel heat sinks allows for a high surface area-to-volume ratio, enabling efficient heat transfer within a compact form factor.

Additionally, the fabrication processes for microchannel heat sinks are increasingly sophisticated and economically viable, making them attractive for a wide range of commercial applications. The focus of this study is to investigate the cooling of electronic chips using forced convection within silicon-based single microchannel heat sinks. Using computational fluid dynamics (CFD) enables a detailed analysis of fluid flow and heat transfer characteristics within these microchannel heat sinks, offering valuable insights into their thermal performance. Understanding these characteristics is essential for optimising the design and operation of microchannel heat sinks to dissipate heat generated by electronic chips effectively. This study uses the commercial CFD software FLUENT to simulate the fluid flow and heat transfer processes within single microchannel heat sinks. The computational domain is discretised using a combination of non-uniform grids on the flow face and uniform grids along the flow direction. Grid generation is facilitated by Gambit software, incorporating the Cooper method for three-dimensional grid generation. Various parameters, including pressure, velocity, and temperature distributions at the inlet and outlet, are analysed to evaluate the thermal performance of the microchannel heat sinks.

Furthermore, this study examines the effect of different flow rates on heat transfer characteristics and variations in pressure drop across the microchannel heat sink. The simulation employs the Semi-Implicit Method for Pressure-linked Equations (SIMPLE) with a second-order upwind scheme to model fully-developed laminar Flow. Continuity, momentum, and energy equations are solved separately, ensuring accuracy in predicting fluid flow and heat transfer behaviours within the microchannel heat sink.

2. LITERATURE SURVEY

The utilisation of nanofluid became a goal for many researchers. The newly created family of ultrafine nanoparticle coolants (1–100 nm) showed stimulating behaviour in lab trials, offering increased heat transfer characteristics compared with pure fluids [1, 2–4]. Xu et al. [5] explored analytical methods for determining heat transmission and pressure decreases for the heat sink in the parallel plate. They minimised the entropy generation rate for fine geometry optimisation and flow conditions. Ultimately, the effects of the numerical model and the heat sink were utilised to determine air conditioning limits. Water cooling technology has been researched by Ellsworth et al. [6], using hybrid air for water cooling in the old IBM systems, then passive water cooling. Beware how and why water refrigeration has been carried out to provide the cooling power required while preserving easy operation at the module level. The enhancement in heat transfer characteristics and properties of liquid Flow of rectangular micro-channel heat sinks (MCHS) have been examined numerically by Raghuraman et al. [7]. Water was the working fluid. The flow domain was discretised and solved using commercially available CFD code, ANSYS CFX 14.5. The Al2O3 / water nanofluid's laminar convective coefficient was studied experimentally and numerically in the circular tube by Azari et al. [8] under identical conditions on the surface.

Three specific modules have been developed: the first was a single-phase fixed physical properties model, the second was a single-phase variable physical properties model, and the third was a double-phase model of discrete particles. The experimental heat transfer and pressure drop characteristic linked with the MCHS was examined by Ali et al. [9]. As a base fluid, TiO2 nanofluid was employed as a coolant at 15% weight concentrations in water, and its efficiency was compared to 100 W, 125 W and 150 W distilled water. The results showed that the thermal efficiency of TiO2 nanofluid relied greatly on the heat flux and that its utility could be done more effectively at a lower thermal cost. The angle effect on the pine-fin heat sink channel was further examined by employing water-based graphene (GNP) nanofluids in the 0.25–0.75 LPM. Al Rashed et al. [10] evaluated the effect of nanoparticles on the effectiveness of a microprocessor used for CPU cooling. Both experiments and numerical analysis were carried out for distilled water at the beginning, then for nanofluid CuO-water with 2 different Ø of 0.86 and 2.25 vol.%.

The numeric application of nanofluids in micro-pin-fin heat sinks was studied by Seyf et al. [11]. 3-D steady energy equation and Navier-Stokes equation were modelled with a finite volume approximation, iteratively solved using a SIMPLE algorithm, and the heat transfer behaviour in Micro-Pin-Fin Heat Sinks. Jajja et al. [12] examined systemic effects on the base temperature of the microprocessor of the geometry sink with cooling fluid (water). They developed a high-heat microprocessor using a block of copper with a cylindrical shape and a power of 325 W, which was compared with commercially available heat sinks and nanofluids. The potential of thermal sink shapes was investigated, and it was found that they are enough to lower high-temperature generated microprocessors to safe and acceptable values. Panchal [13] studied the thermal characteristics of a prismatic pouch battery with LiFePO4 electrode composition and components. Experimental studies are utilised to characterise the battery, allowing the production of an analytical battery temperature model for vehicle models. Electrical evidence is also provided to validate electrochemistry-based battery thermal models. Jilte et al. [14] developed a new enhanced battery module arrangement employing two-layer nanoparticle improved phase change materials. The scheme recommended m × n × p organised where m indicated the number of Li-ion 18,650 cells, and n and p denoted the number of primary and secondary containers. Every Li-ion cell was permitted to release at 3 circumstances for two alternative arrangements: 7 × 7 × 1 and 7 × 1 × 1. Kabeel et al. [15] conducted a numerical study using the Icepak 4.2.8 programme to assess the air cooling performance of an electronic cabinet incorporating heat sources (electronic circuit boards) using an axial fan. Shah et al. [16] examined the thermal conductivity and viscosity of produced α-alumina nanofluids. Their studies demonstrated that α-Al2O3 nanofluid was more thermally stable than typical cooling liquids. Sultan [17] explored the heat transfer increase within projecting heat sources. He also obtained a relation for the Nu as a function of the Richardson number. Esmaeil et al. [18] carried out studies for mixed convection heat transfer from a heating source chilled by forced nanofluid Flow. Wiriyasart et al. [19] studied experimentally the thermal cooling enhancement approach of 2 microprocessors workstation PC with an air cooler unit. Because of the space lack and heat created in PCs and electronic equipment, heat pipes are ideal for cooling microprocessors, Siricharoenpanich et al. [20]. Affecting parameters, such as heat source inclination angle, the presence of porous material, employed fluids, and temperature change, were put into worry. Jawad et al. [21] employed an aluminium chip, nano-silicon carbide (SiC) tubes, and paraffin wax to improve the heater properties. Experimental results revealed the new air heater's adaptability to run under Bagdad's weather patterns.

3. METHODOLOGY

The three-dimensional fluid flow and heat transfer in a rectangular microchannel heat sink are analysed using water as the cooling fluid. A schematic of the structure of a rectangular microchannel heat sink is depicted in Figure 1. The micro-heat sink model comprises a 10 mm long (L=10 mm) silicon wafer with a width of w = 57 µm, a depth of h = 180 µm, and walls separated by a thickness of 43 µm. Consistent heat flux is applied to the bottom surface of the heat sink. Heat transmitted in the unit cell is a conjugate problem which combines heat conduction through the material and dissipates away by convection of the cooling fluid in the microchannel.

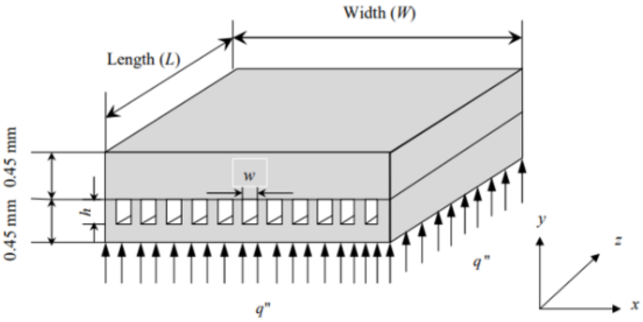

Figure 1. Schematic of the microchannel heat sink

A rectangular microchannel with dimensions of 10 mm × 100 µm × 900 µm is considered. Each channel has a height (h), length (L), and width (w). The bottom surface of the heat sink at y = 0 is uniformly heated with constant heat flux, while the top surface at y = H is adequately insulated. Additionally, adiabatic conditions are applied at the other limits. The fluid flowing through the channel enters at a temperature of 20°C due to a pressure loss of 50 kPa. The direction of the fluid flow is parallel to the z-axis, as depicted in Figure 2. The Flow is assumed to be laminar and hydrodynamically and thermally fully developed. Furthermore, the thermophysical properties are expected to remain constant. The geometric dimensions of the microchannels, including the hydraulic diameter determined from the equation, are listed in Table 1.

4. RESULT AND DISCUSSION

A three-dimensional model is constructed to examine the Flow and conjugate heat transfer in microchannel-based heat sinks for electronic packaging applications. FLUENT is employed for numerical calculations, and the results demonstrate the effects of temperature distribution, heat flux distribution, and the average heat transfer coefficient in the microchannel heat sinks. The average area of the microchannel is calculated by averaging the total number of nodes along the periphery of the inner wall to plot different graphs.

Table 1: Geometric dimensions of the single microchannel.

H	h	W	w	S_t	S_b	t	L	D_h
900	180	100	57	450	270	21.5	10	86.58
All Values in µm								

This study aims to enhance the understanding of fluid flow and heat transfer characteristics in silicon-based microchannel heat sinks designed for electronic cooling. These microchannel heat sinks, with non-circular channels and a silicon base, combine properties such as excellent material compatibility, high surface area per unit volume ratios, and considerable heat transfer performance potential, all achievable through sophisticated yet cost-effective fabrication procedures. These features make silicon-based microchannel heat sinks particularly appealing for various commercial applications. Various pressure drops, as outlined in Table 2, are applied in FLUENT for simulation purposes.

Table 2: Flow conditions for microchannel heat sink

Sink Material	Δp (Pascal)
Silicon	10000
	20000
	30000
	40000
	50000
	60000

The fluid enters the microchannel at a pressure of 50 kPa with a constant inlet temperature of 20°C. After passing through the channel, the fluid is discharged to the atmosphere, i.e., gauge pressure is considered. A continuous heat flux of q" = 90 W/cm² is applied to the bottom wall of the heat sink. The pressure contours inside the channel are illustrated in Figure 2.

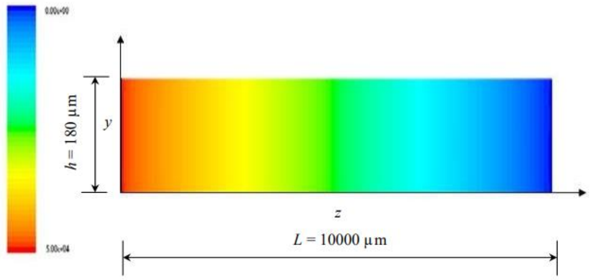

Figure 2. Pressure contours of channel in y-z plane at x = 50 µm for Δp = 50 kPa and q"= 90 W/cm²

The temperature of the fluid at the inlet is initially uniform at 20°C. The displayed temperature profiles result from assuming a hydrodynamically fully developed flow. The microchannel heat sink's solid and fluid sections have a temperature rise along the flow direction. It is worth noting that the temperature point is situated on the heated base surface of the heat sink, precisely below the channel outlet, as illustrated in Figure 3.

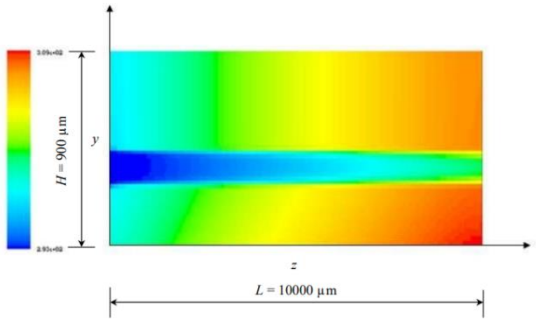

Figure 3. Temperature contours in y-z plane at x = 50 µm for Δp = 50 kPa and q"= 90 W/cm²

This is due to the low velocity of the fluid flow and the resulting high heat flux concentration. Along the flow path, the coolant temperature increases due to heat input, indicating the fluid's maximum temperature at the channel's exit. Inside the channel, the fluid temperature rises from 20°C to 36°C. In order to predict fluid flow and thermal features, the contours of temperature in the fluid at the inlet and outlet of the channel are depicted in Figure 4.

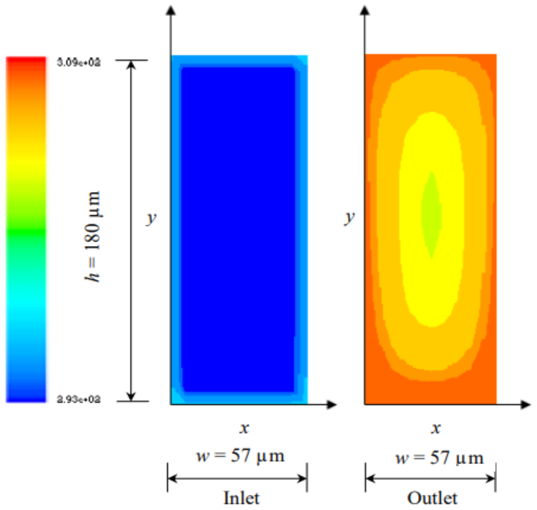

Figure 4. Temperature contours of channel inlet and outlet in the x-y plane at x = 57 µm for Δp = 50 kPa and q” = 90 W/cm^2

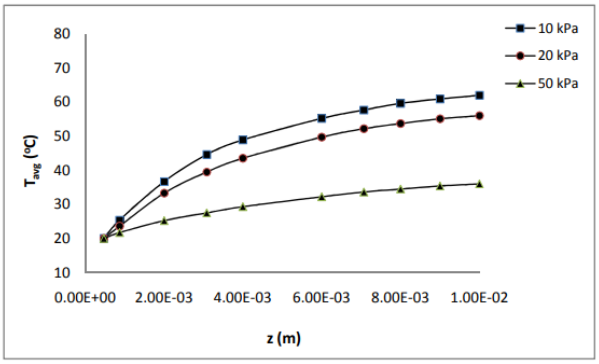

Figure 5: Average liquid temperature variations inside the channel in the x-y plane for different pressure drops and q” = 90 W/cm^2

Inside the microchannel, the temperature of the fluid rises upto 62°C in the case of a 10 kPa pressure drop. This is attributed to the low liquid velocity, causing a large portion of the heat to be conducted into the front part of the heat sink. Conversely, in the case of a 50 kPa pressure drop, the temperature rises upto 36°C due to high liquid velocity, resulting in a smaller portion of heat being conducted into the front part of the heat sink, all while maintaining a constant inlet temperature of 20°C, as shown in Figure 5. The variations in liquid temperature for different pressure drops at a constant heat flux are presented in Table 3.

Table 3: Inlet and outlet temperature of liquid for different pressure drops.

Δp (Pascal)	Inlet temperature (°C)	Outlet temperature (°C)
10000	20	62
20000	20	56
35000	20	46
50000	20	36
65000	20	26

5. CONCLUSIONS

This study highlights the effectiveness of silicon-based microchannel heat sinks in cooling electronic chips through forced convection of water, demonstrating their potential for various commercial applications. We underscore the importance of optimising pressure drops to reduce power consumption while efficiently managing temperature increases within microchannels through precise simulations of fluid flow and heat transfer characteristics. The consistency between our simulation results and existing data validates the reliability of our methodology, offering valuable insights into the design of efficient cooling systems for electronic devices.

REFERENCES

[1]. A. Kamyar, R. Saidur, M. Hasanuzzaman, Application of Computational Fluid Dynamics (CFD) for nanofluids, Int. J. Heat Mass Transf. 55 (15–16) (2012) 4104–4115.

[2]. M. Raja, R. Vijayan, P. Dineshkumar, M. Venkatesan, Review on nanofluids characterisation, heat transfer characteristics and applications, Renew. Sustain. Energy Rev. 64 (2016) 163–173.

[3]. S.M. Sohel Murshed, C.A. Nieto de Castro, A critical review of traditional and emerging techniques and fluids for electronics cooling, Renew. Sustain. Energy Rev. 78 (2017) 821–833.

[4]. W. Daungthongsuk, S. Wongwises, A critical review of convective heat transfer of nanofluids, Renew. Sustain. Energy Rev. 11 (5) (2007) 797–817.

[5]. G. Xu, B. Guenin, M. Vogel, Extension of air cooling for high power processors, Thermomech. Phenom. Electron. Syst. - Proceedings Intersoc. Conf. 1 (858) (2004) 186–193.

[6]. M.J. Ellsworth, L.A. Campbell, R.E. Simons, M.K. Iyengar, R.R. Schmidt, R.C. Chu, The evolution of water cooling for IBM large server systems: back to the future, in 2008 11th IEEE Intersoc. Conf. Therm. Thermomechanical Phenom. Electron. Syst. I-THERM, 2008, pp. 266–274.

[7]. D.R.S. Raghuraman, R. ThundilKaruppa Raj, P.K. Nagarajan, B.V.A. Rao, Influence of aspect ratio on the thermal performance of rectangular shaped microchannel heat sink using CFD code, Alexandria Eng. J. 56 (1) (2017) 43–54.

[8]. A. Azari, M. Kalbasi, M. Rahimi, CFD and experimental Investigation on the heat transfer characteristics of alumina nanofluids under the laminar flow regime, Brazilian J. Chem. Eng. 31 (2) (2014) 469–481.

[9]. H.M. Ali, W. Arshad, Effect of channel angle of the pin-fin heat sink on heat transfer performance using water-based graphene nanoplatelets nanofluids, Int. J. Heat Mass Transf. 106 (2017) 465–472.

[10]. MH Al-Rashed, G. Dzido, M. Korpy´s, J. Smołka, J. Wojcik, Investigation on the CPU nanofluid cooling, Microelectron. Reliab. 63 (2016) 159–165.

[11]. H.R. Seyf, M. Feizbakhshi, Computational analysis of nanofluid effects on convective heat transfer enhancement of micro-pin-fin heat sinks, Int. J. Therm. Sci. 58 (2012) 168–179.

[12]. S.A. Jajja, W. Ali, H.M. Ali, A.M. Ali, Water cooled minichannel heat sinks for microprocessor cooling: effect of fin spacing, Appl. Therm. Eng. 64 (1–2) (2014) 76–82.

[13]. S. Panchal, “Impact of vehicle charge and discharge cycles on the thermal characteristics of lithium-ion batteries,” 2014.

[14]. R. Jilte, A. Afzal, S. Panchal, A novel battery thermal management system using nano-enhanced phase change materials, Energy 219 (2021), 119564.

[15]. A.E. Kabeel, A.M. Khalil, G.I. Sultan, M.I. El-Hadary, CFD Modeling of the effect of the air-cooling on electronic heat sources, in International Conference of Fluid Dynamics 11 ICFD11-eg- 4060, 2013, pp. 1–6.

[16]. J. Shah, M. Ranjan, V. Davariya, S.K. Gupta, Y. Sonvane, Temperature-dependent thermal conductivity and viscosity of synthesised a-alumina nanofluids, Appl. Nanosci. 7 (8) (2017) 803–813.

[17]. G.I. Sultan, Enhancing forced convection heat transfer from multiple protruding heat sources simulating electronic components in a horizontal channel by passive cooling, Microelectron. J. 31 (9) (2000) 773–779.

[18]. K.K. Esmaeil, G.I. Sultan, F.A. Al-Mufadi, R.A. Almasri, Experimental heat transfer from heating source subjected to rigorous natural convection inside the enclosure and cooled by forced nanofluid Flow, J. Heat Transfer 141 (7) (2019) 1–9.

[19]. S. Wiriyasart, C. Hommalee, P. Naphon, Thermal cooling enhancement of dual processors computer with thermoelectric air cooler module," case studies, Therm. Eng. 14 (2019).

[20]. A. Siricharoenpanich, S. Wiriyasart, A. Srichat, P. Naphon, Thermal management system of CPU cooling with a novel short heat pipe cooling system," case studies, Therm. Eng. 15 (2019), 100545.

[21]. QA Jawad, A.M.J. Mahdy, A.H. Khuder, M.T. Chaichan, Improve the performance of a solar air heater by adding aluminium chip, paraffin wax, and nano-SiC," case studies, Thermal Engineering, 19 (2020).