

Numerical Optimization of a Developed Multi-Baffle Heat Sink for Modern Electronic Devices

Iman H. Alwan^{1,*} , Mohammed Kh. Abbas¹ 

¹Mechanical Engineering Department, College of Engineering, Diyala University, Diyala, 32001, Iraq

mechmsc9231@uodiyala.edu.iq, mohammedkhudhair_eng@uodiyala.edu.iq

Abstract

Optimal thermal control is crucial for ensuring the efficiency and stability of modern electronic devices. This paper analyzes four multi-barrier heat sink models using computational fluid dynamics (CFD) to optimize heat removal from a hot surface under constant convection. We analyzed the models under laminar flow conditions in ANSYS Fluent, using five inlet velocities in a low-Reynolds-number range. We used the heat transfer coefficient and Nusselt number as key performance indicators. The numerical results showed that adding inclined barriers improves coolant distribution and enhances fluid mixing within the flow channels, thereby increasing the localized convection heat transfer rate. Among the designs studied, configuration four model (M4) exhibited the highest thermal performance, achieving the greatest increase in both the heat transfer coefficient 1057.24 W/m²K and the Nusselt number, 9.06. The improvement is more pronounced at higher Reynolds numbers, demonstrating the positive effect of flow direction on convection heat transfer. Overall, the proposed multi-barrier design provides an efficient way to improve passive cooling without additional power consumption or system complexity, making it a promising approach for integrated electronics cooling applications. This study provides useful design indicators for developing heat sinks used in modern electronic devices.

Keywords: Flow simulation, heat sink, heat transfer, Nusselt number

Article history: Received: 19 May 2026, Accepted: 30 Aug 2026, Published: 15 Sep 2026.

This article is open access under the CC BY 4.0 license (<http://creativecommons.org/licenses/by/4.0/>).

1. Introduction

Hot-surface temperature and heat-distribution uniformity are key factors that determine the thermal performance and service life of heat sinks used in modern power and electronic equipment. Large temperature variations can cause thermal stresses, uneven material expansion, and localized corrosion, negatively affecting the reliability of thermal systems in both small- and large-scale industrial applications [1]. These phenomena are typically associated with poor flow uniformity within conduits, inconsistent heat transfer, or localized heat sources. Therefore, achieving better thermal uniformity and minimizing hot-surface temperature variations is a primary objective in

designing heat sinks for cooling high-temperature electronic components.

The scientific literature has focused on this topic by studying and optimizing integrated thermal systems, such as microchannel cooling systems and high-surface-area structures. These systems include microchannel heat sinks [2] and [3], reactive heat exchangers [4] and [5], plate heat exchangers [6] and [7], and fuel-cell cooling systems [8, 9]. Uniform flow distribution within parallel channels can reduce the temperature difference at the hot surface, improve the heat transfer coefficient, and minimize pressure loss.

Flow maldistribution is a major engineering challenge in multichannel heat sinks, and researchers have studied this phenomenon

extensively in recent years [10–12]. Liu et al. [10] showed that flow maldistribution in plate-based cooling systems leads to significant temperature variation in fuel cells. Wang et al. [11] analyzed how inlet and outlet collector design affects fluid pressure and distribution within the cooled plates. In his research, Wang [12] improved the flow fields in single-channel cooling systems by using geometric coefficients as optimization variables.

The literature has described several strategies for improving fluid distribution using two-dimensional diffusers and geometrically modified structures [13–15]. Lu et al. [13] proposed a structural approach that integrates heat exchangers and microreactors to improve inlet fluid distribution and reduce flow concentration in certain paths. Fan et al. [14] tested a plate diffuser to measure flow homogeneity, showing the potential to achieve a more balanced distribution with significantly reduced energy loss.

Liu et al. [15] investigated geometric channel branching and the effect of dividing a single channel into two sub-channels, finding that appropriate branching patterns can mitigate thermal differentials between channels and improve cooling performance in parallel-channel systems. Other studies have focused on the effects of collector shape, orifice distribution, and variations in inlet/outlet position. Zhang et al. [16] showed that modifying the collector shape reduces flow variability and improves thermal distribution. Cyril et al. [17] demonstrated that using stepped or tapered diffusers can achieve uniform distribution in low-velocity flows.

Kumaran et al. [18] presented a numerical comparison of various collector shapes, including I, C, V, Z, and U, showing that some shapes offer better thermal performance under different operating conditions. Wang et al. [19] observed that a U-shaped arrangement in heat exchangers achieves greater uniformity compared to a Z-shaped arrangement under single-phase flow, and Zhang et al. [20] experimentally confirmed these results under various operating conditions.

In addition to modifying collectors, researchers have used baffles to redirect flow and reduce

velocity-distribution variations within channels. Liu et al. [21] showed that incorporating small baffles into cooling channels improves thermal uniformity and redirects fluid toward terminal channels that often suffer from poor cooling.

Ahmed et al. [22] showed that branch channels reduce hot spots in liquid-cooled systems, while Wen et al. [23] demonstrated improved distribution by incorporating perforated baffles into collectors. Other studies have demonstrated the effectiveness of porous baffles [24] and cylindrical inserts [25] in improving flow distribution without a significant increase in pressure loss, while Lu et al. [26] developed geometric design algorithms to select the optimal baffle arrangement. Chai et al. [27] and [28] investigated the effect of internal ribs on thermal-hydraulic performance, and their results showed that the ribs prevent a decrease in the heat transfer coefficient along the flow direction.

Despite advances in this field, achieving optimal thermal homogeneity in parallel-channel heat sinks remains a significant engineering challenge, particularly in laminar-flow systems, where mixing enhancement is relatively limited. Common metrics for evaluating heat sink performance include temperature difference (ΔT), pressure loss (ΔP), heat transfer coefficient (h), and Nusselt number (Nu), which together provide a comprehensive picture of heat removal efficiency and flow rate. Achieving a suitable balance between optimizing thermal performance and minimizing pressure loss is essential for electronics cooling applications.

Numerical simulations based on computational fluid dynamics (CFD) have significantly contributed to the study of these phenomena, as they can analyze complex channels and evaluate the impact of geometric modifications at lower cost than experimental testing [29, 30]. Water is typically used as the coolant in these applications because of its high thermal properties, and aluminium is often used as the heat sink material due to its high thermal conductivity and fabricability. Furthermore, the system in this study operates in a low Reynolds number (R) range of laminar flow, making the role of baffles in flow redistribution more critical than in turbulent or high-flow systems.

In this context, this research presents a numerical study using ANSYS Fluent v.15 to analyze the effect of baffles on the thermal performance of a heat sink designed to cool electronic components. The analysis used laminar water flow within parallel aluminium channels, applied a uniform heat flux at the hot surface, and examined geometric models of the graded baffles. Performance was evaluated using the variables temperature difference (ΔT), pressure loss (ΔP), heat transfer coefficient (h), and Nusselt number (Nu) to clarify the relationship between improved flow distribution and reduced temperature difference. The results of this work also aim to contribute to the field of thermal design for integrated heat sinks with digital outputs that can be used both technically and engineering-wise in high-temperature devices such as modern electronic processing units [31–34].

2. Problem Description

2.1 Geometric Configurations

This research uses numerical analysis to study the flow behavior and heat transfer within a multi-barrier heat sink. All simulations were performed using ANSYS Fluent v.15 software, given its high accuracy and reliability in modelling electronic cooling problems.

A three-dimensional geometric model of the heat sink was constructed to represent the channels and barriers fully. We assumed three-dimensional, laminar, incompressible flow, with constant fluid and solid physical and thermal properties within the studied temperature range. Fig. 1 shows the geometric configurations of the four heat sink models, and Table 1 lists all dimensions of the M1, M2, M3, and M4 used in this study.

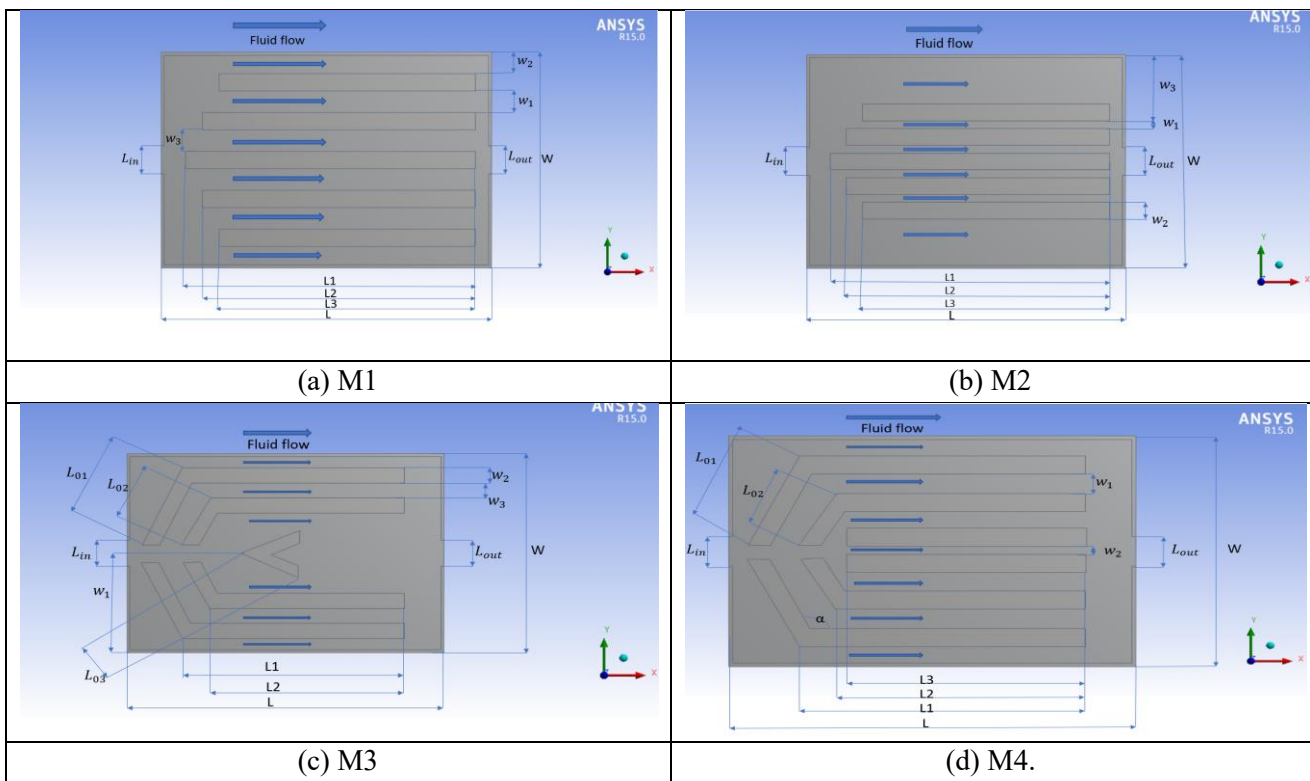


Fig. 1. Geometric configurations of the four heat sink models.

Table 1: Geometric parameters of the multi-baffle-type heat sink (MBHS).

Model	L/W/H(mm) H1/Lin/Lout	Ln(mm) (n=1,2,3)	Wn(mm) (n =1,2,3)	L0n(mm)	α (°)
M1	200/140/5 3/18/18	175/165/155	14.17/14.17/14.17	—	—
M2	200/140/5 3/18/18	175/165/155	5/11/41.5	—	—
M3	200/140/5 3/18/18	140/122	70/10/11	60/36/40	115°

M4	200/140/5 3/18/18	140/122/118	10/5	60/36	115°
----	----------------------	-------------	------	-------	------

2.2 Mathematical Model

Water was used as the cooling fluid, with an inlet temperature of 298 K and an inlet velocity range of 0.06 - 0.28m/s. At the outlet, we applied a pressure condition of 0 Pa to allow the flow to exit without further restrictions. We applied a constant heat flux of 58,000 W/m² to the heat sink's bottom surface. In contrast, we considered the remaining surfaces thermally insulated, a common practice in electronic device cooling studies. Table 2 lists the entry velocities and Reynolds numbers used in the numerical simulation.

Table 2: Reynolds numbers corresponding to the heat sink inlet fluid velocities (u).

u(m/s)	0.06	0.12	0.17	0.23	0.28
Re	346	690	980	1300	1600

The results indicate that the highest Reynolds number obtained was approximately 1600 at a flow velocity of 0.28 m/s. These values demonstrate that the flow remained entirely within the laminar flow range (Re < 2300) throughout the controlled operating conditions.

2.3 Governing Equations:

The simulation is based on fundamental equations governing flow continuity, momentum, and heat transfer. These equations can be expressed as follows:

Continuity Equation:

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

Momentum Equation:

$$\frac{\partial(\rho u_i u_j)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\mu \frac{\partial u_i}{\partial x_i} \right) - \frac{\partial p}{\partial x_i} \quad (2)$$

Energy Equation:

$$\frac{\partial(\rho u_i T)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\frac{k}{c_p} \frac{\partial T}{\partial x_i} \right) + S_T \quad (3)$$

For the solid

$$\frac{\partial}{\partial x_i} \left(k_s \frac{\partial T}{\partial x_i} \right) = 0 \quad (4)$$

The basic heat-sink geometry is prepared in ANSYS Design Modeler v.15, and the numerical mesh is generated in ANSYS Meshing. Numerical solutions are performed using ANSYS Fluent v.15, a standard computational fluid dynamics tool. The

study uses an irregular mesh, which is checked at the inlet, outlet, and channel walls to improve variable accuracy.

Boundary Conditions Used in the Simulation:

1. The fluid used is water, and the multi-barrier heat sink (MBHS) is made of aluminium.
2. A constant heat flux of $q = 58000 \text{ W/m}^2$
3. The fluid inlet velocity (shown in Table 2) is constant, while the inlet temperature is $T_{in} = 298 \text{ K}$.
4. The exit condition in the software was determined by setting a static pressure of 0 Pa at the outlet.

2.4 Definition of Basic Parameters

To analyze the flow characteristics more precisely, the Reynolds number used is defined as follows:

$$Re = \frac{\rho u D_h}{\mu} \quad (5)$$

Where u: water velocity, μ : Dynamic viscosity, D_h : Hydraulic diameter of the inlet, and ρ : fluid density.

The heat-transfer coefficient (h) is

$$h = \frac{q}{\Delta T} = \frac{q}{T_h - T_m} \quad (6)$$

$$T_m = (T_{in} + T_{out})/2 \quad (7)$$

Nusselt Number (Nu) is:

$$Nu = \frac{h D_h}{k} \quad (8)$$

3. Numerical Simulation

3.1 Grid Independence

In the numerical simulation, we chose the M4 grid to verify numerical-solution independence because it represents the most complex geometry among the investigated models. If mesh independence is achieved for the most complex configuration, the remaining simpler models can also be simulated with sufficient numerical accuracy. We performed a grid independence test using grids with 1509480, 2803109, 3451233, 4152708, and 5600456 cells. The errors between the obtained temperature and pressure drop values at the heating surface were sufficiently small to establish the solution's independence. Table 3 verifies grid independence. To reduce computational costs and save time, we adopted a grid system of 3451233 grids for

subsequent calculations. Fig. 2 shows the final computational mesh corresponding to this selected grid.

Table 3. Grid Independence Test Results

Grid number	$\Delta P(\text{Pa})$	Relative error	$\Delta T(\text{K})$	Relative error
1509480	141.29	16.96 %	71.42	14 %
2803109	167.37	1.63 %	61.97	0.820 %
3451233	170.14	—	62.48	—
4152708	170.46	0.18 %	62.49	0.016 %
5600456	170.00	0.11 %	62.60	0.19 %

After determining the appropriate grid through grid-independence testing, we created an irregular grid with elements concentrated near walls and barriers to capture steep velocity and temperature gradients more accurately.

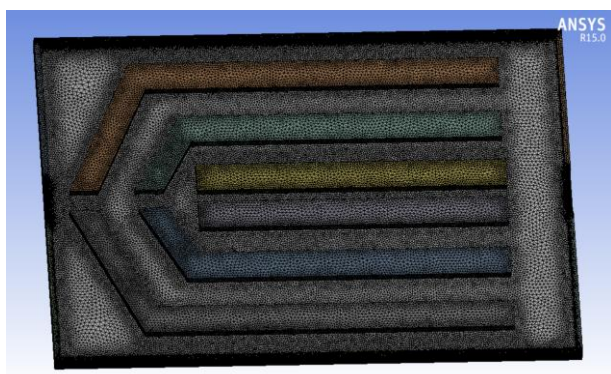


Fig. 2 The final computational mesh for the heat sink M4 after checking the mesh independence.

4. Results and Discussion

4.1 Velocity distribution maps at the midplane of the heat sink at $Re = 1600$

Fig. 3 shows velocity distribution maps at the mid-plane of the heat sink for four different geometries at Reynolds number 1600. In model M1 (a), the flow is mainly concentrated in the mid-channels, with higher velocities than in the lateral channels. This indicates an uneven fluid distribution due to the lower mid-channel resistance. Model M2 (b) shows a relative improvement in the velocity distribution across the channels, as the geometric modification of the flow paths directs a larger proportion of the

fluid towards the terminal channels, although significant local velocity variation remains.

In contrast, models M3 (c) and M4 (d) clearly demonstrate the effect of introducing internal barriers. These barriers force the flow to redistribute within the heat sink, reducing velocity concentration in the mid-channels and promoting more uniform flow across channels. In model M4 (d), the velocity distribution becomes more uniform across the channels, indicating reduced flow maldistribution and a better balance of path resistances. This improved flow regularity enhances the heat sink's thermal performance by increasing heat-transfer efficiency and reducing the likelihood of hot spots, consistent with the expected physical behavior of multichannel heat sinks equipped with internal barriers.

4.2 Pressure distribution maps at the mid-plane of the heat sink at $Re=1600$

Fig. 4 shows the pressure distribution maps at the mid-plane of the heat sink for the four models at $Re=1600$. The figure illustrates how geometric design differences between the models affect the pressure distribution within the channels, showing clear differences between the inlet areas and the rest of the heat sink. This reflects the difference in flow resistance between the studied designs.

4.3 Temperature distribution maps at the mid-plane of the heat sink at $Re=1600$

Fig. 5 shows the temperature distribution maps at the mid-plane of the heat sink for the four models at Reynolds number 1600. Fig. 5 illustrates how channel and barrier geometry affect heat distribution within the heat sink, revealing clear differences in temperature uniformity among the studied models.

4.4 Temperature distribution maps of the bottom plate of the heat sink at $Re=1600$

Fig. 6 shows the temperature distribution maps on the bottom base of the heat sink for the four models at Reynolds number 1600. Fig. 6 illustrates how different channel and barrier designs affect base-temperature uniformity, with color differences indicating clear variations across the studied models.

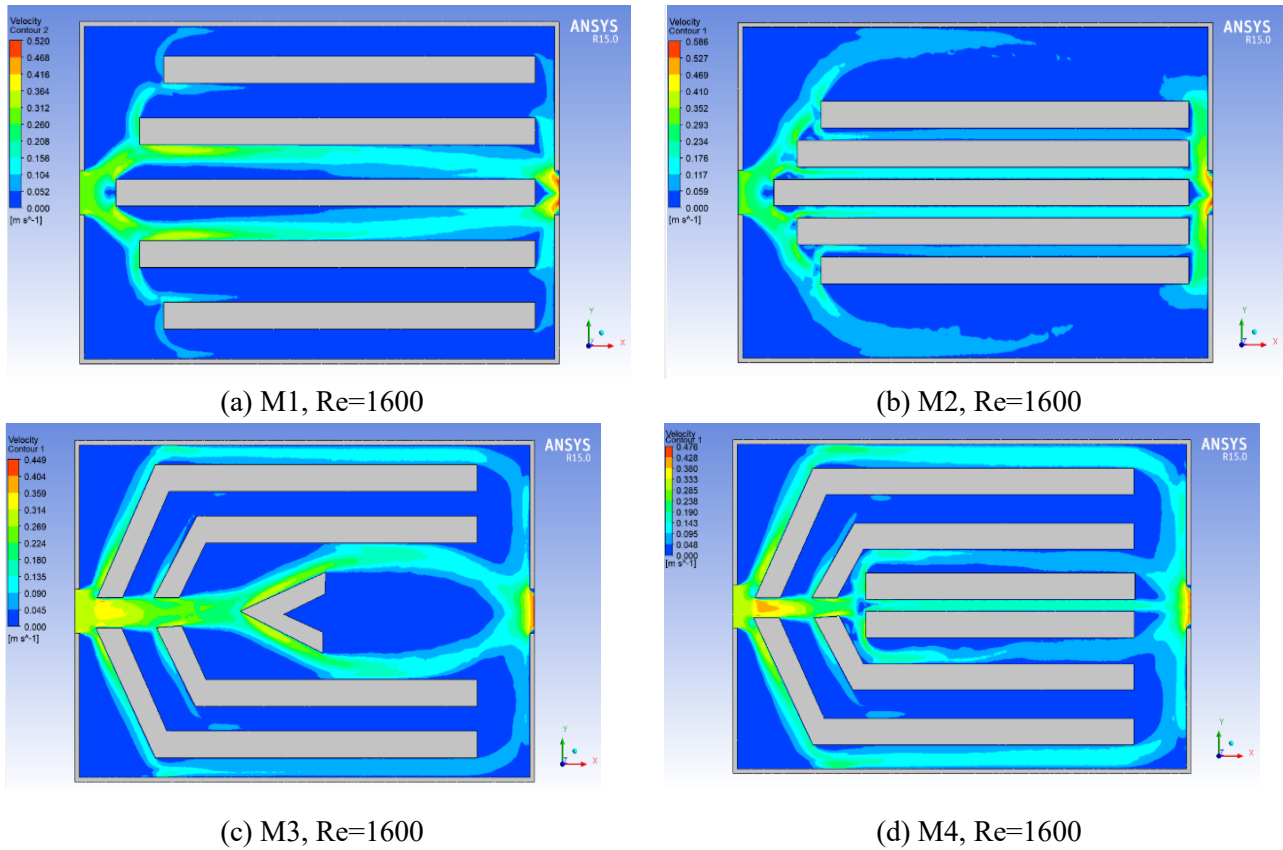


Fig. 3. Velocity at the middle level of the heat sink for the four models at Re = 1600.

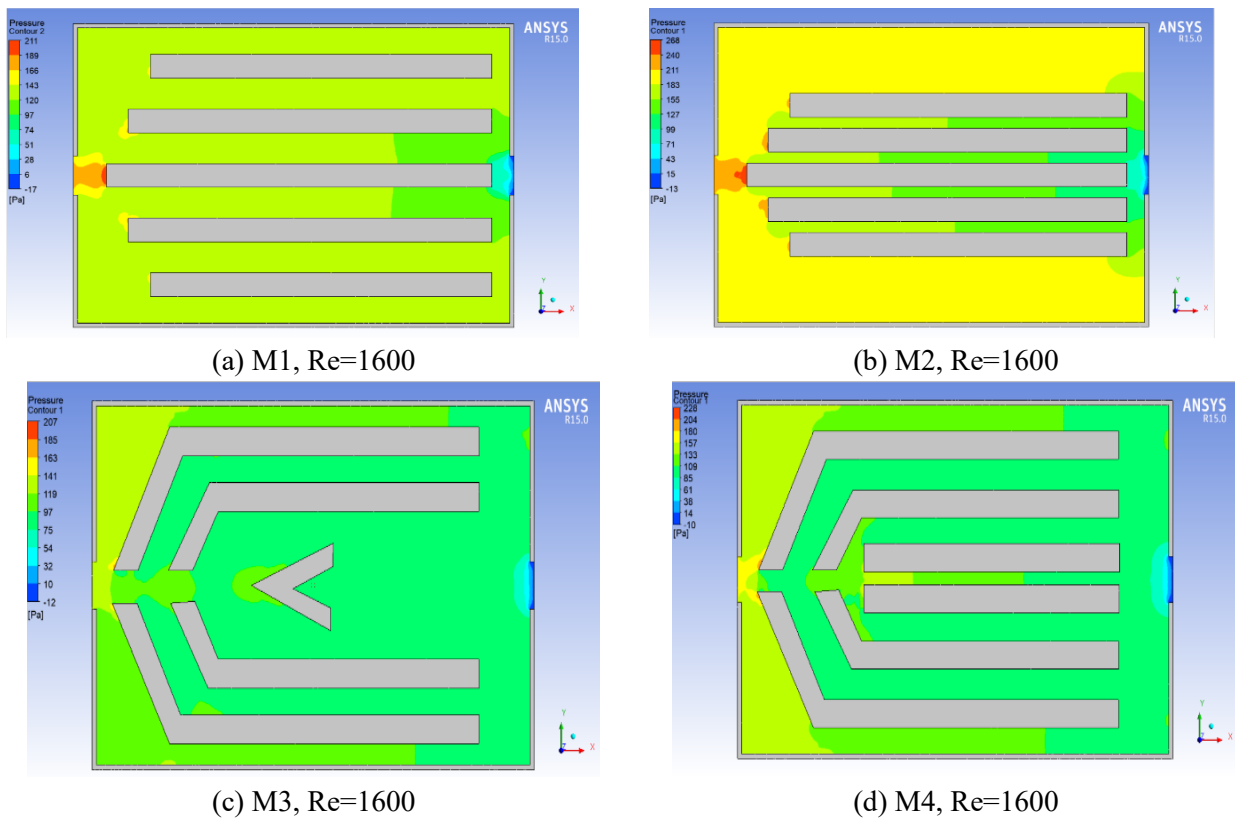


Fig. 4 Pressure contour at the mid-plane of the heat sink at Re = 1600.

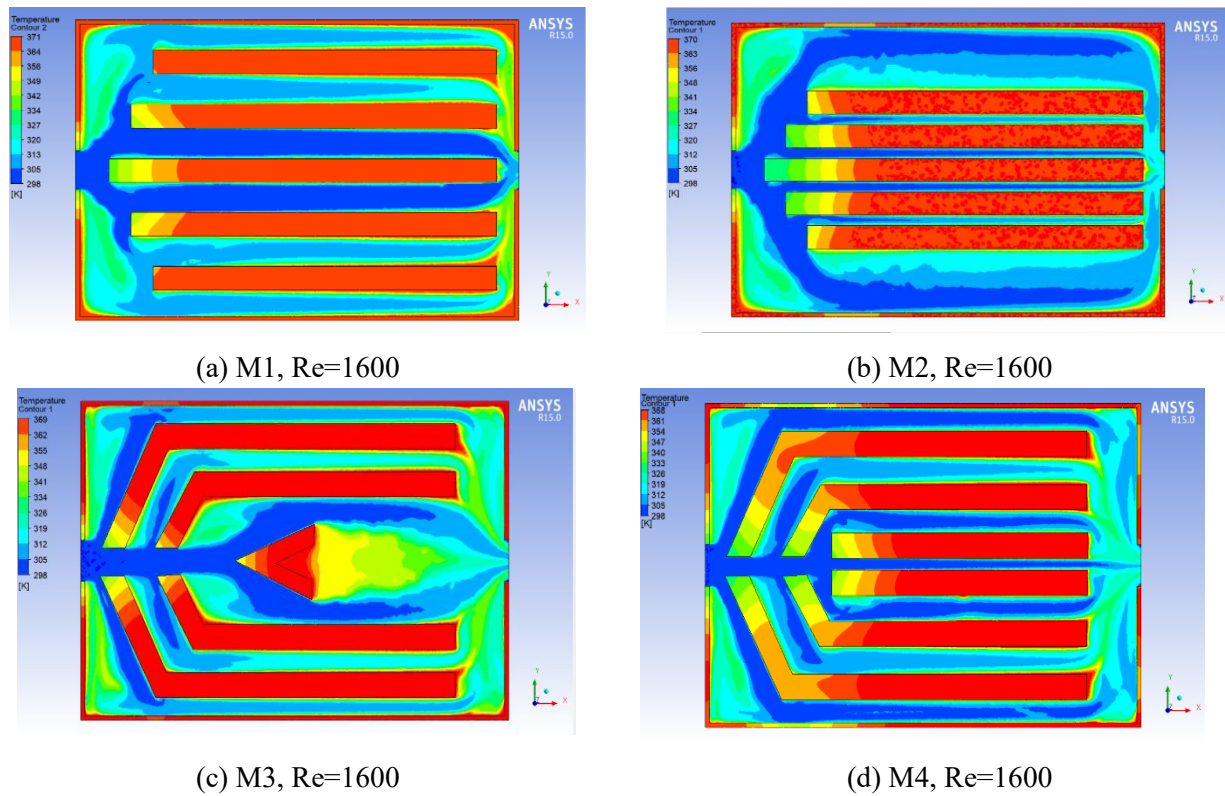


Fig. 5 Temperature contours at the mid-plane of the heat sink at Re=1600.

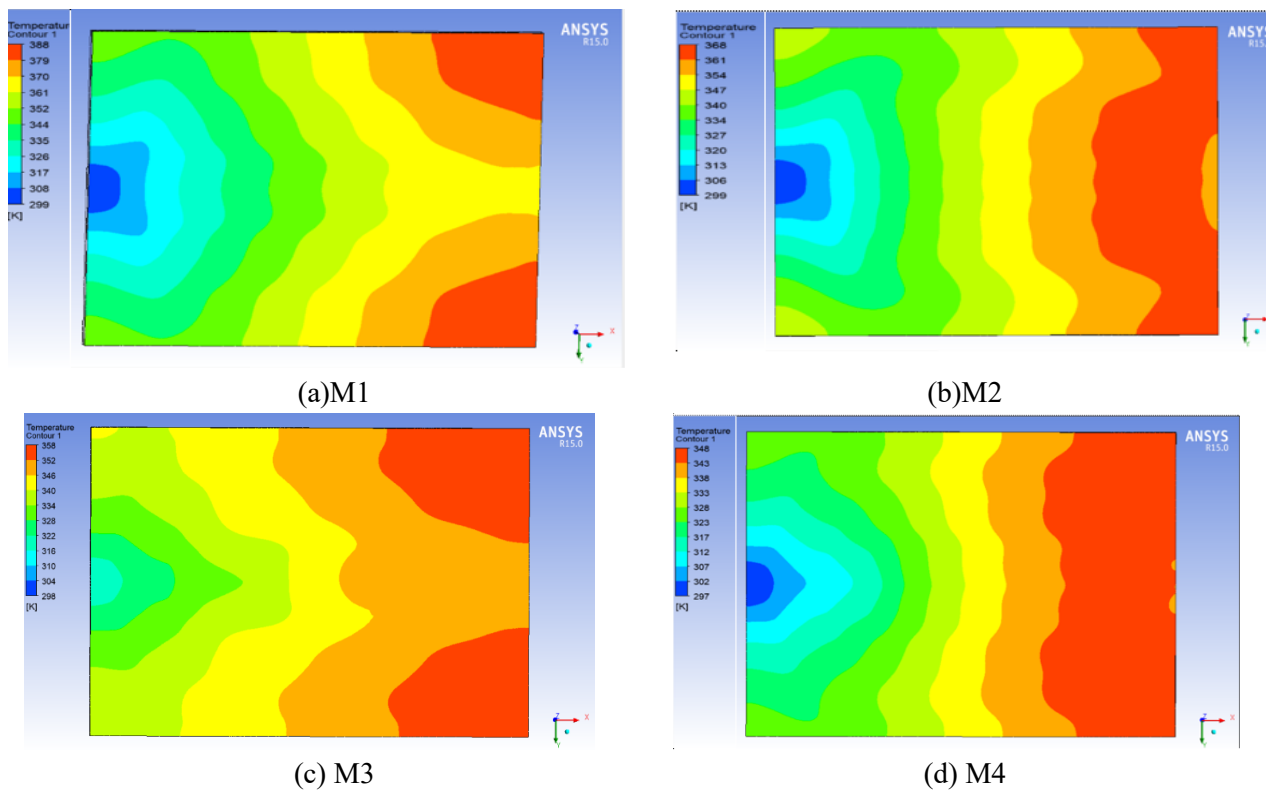


Fig. 6 Temperature contour of the bottom plate of the heat sink at Re=1600.

To evaluate the heat sink's thermal performance, we calculated the heat transfer coefficient and Nusselt number from the surface and fluid temperatures. These indices were used to compare

different configurations and determine the effect of barriers on cooling efficiency and flow uniformity. Table 4 lists ΔP , ΔT , the heat transfer coefficient h ,

and the Nusselt number for models M1–M4 at $Re = 1600$.

Table 4: Comparison of thermal performance parameters for the four heat sink models at $Re = 1600$.

Model	ΔP (Pa)	ΔT (K)	$h(w/m^2 \cdot K)$	Nu
M1	170.37	78.50	738.85	6.33
M2	222.54	75.67	766.49	6.57
M3	139.05	75.80	765.17	6.55
M4	165.8	51	1057.24	9.06

The analysis results showed that model M4 was the most efficient and best-performing of the four models. This model showed the smallest temperature difference, directly indicating its higher efficiency in removing heat from the base surface compared to the other designs. M4 also achieved the highest heat transfer coefficient and the largest Nusselt number, reflecting its engineering design's ability to enhance turbulence within the passages and intensify convection. Despite a relatively high pressure loss, these significant heat gains make model M4 the best and most efficient choice among the models studied.

6. Model Comparison

6.1 The temperatures in the four models

Fig. 7 shows a comparison of the change in temperature difference T_d With the Reynolds number for the four heat sink models. The results show a gradual decrease in T_d . As the Reynolds number increases for all models, it reflects improved heat transfer efficiency due to enhanced convection at higher flow rates. Model M4 exhibits the smallest

temperature difference across the studied range, indicating better thermal performance than the other models.

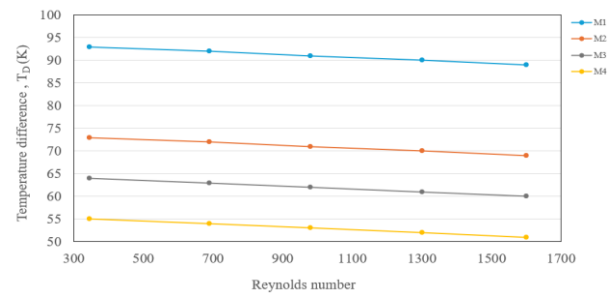


Fig. 7 The temperature difference between the maximum and minimum temperatures in the four models.

6.2 The heat-transfer performance for the four models

Fig. 8 (a) and 8 (b) illustrate the comparison of the heat-transfer coefficient and Nusselt number for the four models in the simulation. As the Reynolds number increases, both the heat-transfer coefficient and the Nusselt number increase. For all Reynolds numbers, the heat-transfer coefficient and Nusselt number of M4 are the largest, which indicates the acceptable heat-transfer performance of M4 due to the lower temperature of the heating surface.

The maximum convective heat transfer for M4 is $1057.24 \text{ W/m}^2 \cdot \text{K}$, and the Nusselt number is 9.06, obtained at the highest evaluated flow rate ($Re = 1600$). These values are the lowest for M1, with a heat-transfer coefficient of $738.85 \text{ W/m}^2 \cdot \text{K}$ and a corresponding Nusselt number of 6.33, as summarized in Table 4

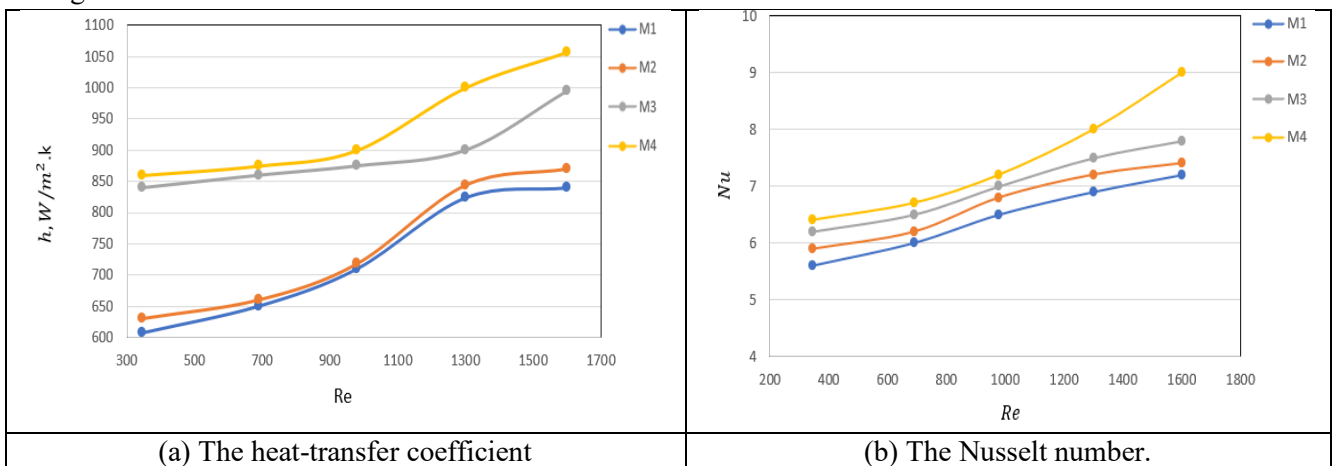


Fig. 8 The heat-transfer performance for the four models.

5. Conclusions

In this paper, we present a novel type of multi-barrier heat sink. We investigated the flow and heat-transfer characteristics of this sink at different inlet velocities through numerical simulations. The results indicate that the heat transfer coefficient of the M4 sink is $1057.24 \text{ W/m}^2\cdot\text{K}$, the temperature difference between the highest and lowest heating surface temperatures is 51 K, and the pressure drop is 165.8 Pa at a Reynolds number of 1600. The simulation results show that the M4 sink exhibits superior heat transfer and flow uniformity compared to the other models studied in this research.

Acknowledgment

The author thanks the College of Engineering, University of Diyala, for their support.

Conflict of Interest

The author declares no conflict of interest.

Nomenclature

H	total height of cooling plate, mm
H_1	height of the counter sink, mm
h	convective heat transfer coefficient, $\text{W/m}^2 \text{ K}$
k_s	thermal conductivity of Al, W/m K
k_f	thermal conductivity of fluid, W/m K
D_h	hydraulic diameter, m
ΔT	average temperature difference, K
T_{in}	inlet temperature, K
T_{out}	outlet temperature, K
T_h	average temperature of heating surface, K
T_d	temperature difference between the maximum and minimum, K
u	velocity, m/s
L	length of heat sink, mm
L_i	length of flow channel, mm
W_i	width of flow channel, mm
L_{oi}	length of baffle, mm
Nu	Nusselt number
ΔP	pressure drop of the heat sink, Pa
P_{out}	outlet pressure, Pa
P_{in}	inlet pressure, Pa
q	heat flux, W/m^2
Re	Reynolds number
T	temperature, K
Greek symbols	
μ	kinematic viscosity, kg/m s
ρ	density, kg/m^3
β	baffles intersection angle, °
Subscripts	
i	index in the x-direction
j	index in the y-direction
n	flow channel number
x,y,z	three coordinates shown in Fig. 1, mm

References

- [1] Kraus, A. D., & Bar-Cohen, A. (1995). "Thermal analysis and control of electronic equipment". McGraw-Hill.
- [2] Kandlikar, S. G. (2005). High flux heat removal with microchannels—A roadmap of challenges and opportunities. *Heat Transfer Engineering*, 26 (8), 5–14. <https://doi.org/10.1080/01457630591003655>
- [3] Tuckerman, D. B., & Pease, R. F. W. (1981). High-performance heat sinking for VLSI. *IEEE Electron Device Letters*, 2 (5), 126–129. <https://doi.org/10.1109/EDL.1981.25367>
- [4] Bird, R. B., Stewart, W. E., & Lightfoot, E. N. (2002). "Transport phenomena" (2nd ed.). John Wiley & Sons.
- [5] Wakao, N., & Kaguei, S. (1982). "Heat and mass transfer in packed beds". Gordon and Breach Science Publishers.
- [6] Thome, J. R. (1990). "Enhanced heat transfer". Hemisphere Publishing Corporation.
- [7] Incropera, F. P., & DeWitt, D. P. (2001). "Fundamentals of heat and mass transfer" (5th ed.). John Wiley & Sons.
- [8] Larminie, J., & Dicks, A. (2003). "Fuel cell systems explained" (2nd ed.). John Wiley & Sons.
- [9] Wang, C. Y. (2004). Fundamental models for fuel cell engineering. *Chemical Reviews*, 104 (10), 4727–4766. <https://doi.org/10.1021/cr020718s>
- [10] Liu, H., et al. (2008). Flow uniformity in parallel channels for fuel cells and chemical reactors. *Journal of Power Sources*, 178 (1), 103–113. <https://doi.org/10.1016/j.jpowsour.2007.11.083>
- [11] Wang, Y., et al. (2009). Flow and pressure distribution in plate heat exchanger-based fuel cell stacks. "International Journal of Heat and Mass Transfer", 52" (25–26), 5507–5516. <https://doi.org/10.1016/j.ijheatmasstransfer.2009.06.046>
- [12] Wang, Y. (2010). Optimization of serpentine cathode flow field using variable channel heights. *Journal of Power Sources*, 195 (9), 2940–2948. <https://doi.org/10.1016/j.jpowsour.2009.11.033>
- [13] Luo, L., et al. (2009). Constructal design of fluid distributors. "International Journal of Heat and Mass Transfer", 52" (15–16), 3558–3564. <https://doi.org/10.1016/j.ijheatmasstransfer.2009.03.013>
- [14] Fan, A., et al. (2010). Experimental investigation of flow distribution in plate-type fluid distributors. *Chemical Engineering Journal*, 173 (2), 456–463. <https://doi.org/10.1016/j.cej.2010.08.005>
- [15] Liu, H., et al. (2010). Symmetric bifurcation flow structures. *Chemical Engineering Science*, 65 (7), 2177–2186. <https://doi.org/10.1016/j.ces.2009.11.028>
- [16] Zhang, K., et al. (2013). Header design for plate-fin heat exchangers. "Applied Thermal Engineering", 50" (1), 220–228. <https://doi.org/10.1016/j.applthermaleng.2012.06.009>

- [17] Cyril, N., et al. (2012). Flow distribution improvement in mini-channel systems. *International Journal of Heat and Mass Transfer*, 55 (13–14), 3533–3544.
<https://doi.org/10.1016/j.ijheatmasstransfer.2012.03.013>
- [18] Kumaran, K., et al. (2014). Influence of header shape on flow uniformity. *International Journal of Thermal Sciences*, 79, 74–85.
<https://doi.org/10.1016/j.ijthermalsci.2014.01.002>
- [19] Wang, G., et al. (2015). Single-phase flow distribution in U- and Z-type heat exchangers. *Applied Thermal Engineering*, 85, 330–338.
<https://doi.org/10.1016/j.applthermaleng.2015.06.024>
- [20] Zhang, Z., et al. (2016). Distributor configurations for plate-fin heat exchangers. *Applied Thermal Engineering*, 94, 371–379.
<https://doi.org/10.1016/j.applthermaleng.2015.10.131>
- [21] Liu, H., et al. (2017). Thermal performance of mini-channel heat sinks with baffles. *Applied Thermal Engineering*, 112, 1430–1440.
<https://doi.org/10.1016/j.applthermaleng.2016.10.071>
- [22] Ahmed, K., et al. (2017). Sub-channel structures for hot-spot mitigation. *Energy Conversion and Management*, 138, 60–70.
<https://doi.org/10.1016/j.enconman.2017.01.031>
- [23] Wen, J., et al. (2018). Improved header design with punched baffles. *Applied Thermal Engineering*, 130, 1460–1471.
<https://doi.org/10.1016/j.applthermaleng.2017.11.033>
- [24] Ismail, Y., et al. (2018). Porous baffle structures for flow distribution. *International Journal of Heat and Mass Transfer*, 127, 1130–1139.
<https://doi.org/10.1016/j.ijheatmasstransfer.2018.08.095>
- [25] Dabiri, S., et al. (2018). Cylindrical obstacles in flow distributors. *Chemical Engineering Research and Design*, 131, 417–427.
<https://doi.org/10.1016/j.cherd.2018.01.016>
- [26] Luo, L., et al. (2018). Heuristic optimization of baffled fluid distributors. *Energy*, 165, 1227–1238.
<https://doi.org/10.1016/j.energy.2018.09.158>
- [27] Chai, X., et al. (2019). Rib geometry effects on thermal-hydraulic performance. *International Journal of Heat and Mass Transfer*, 129, 327–339.
<https://doi.org/10.1016/j.ijheatmasstransfer.2018.09.023>
- [28] Chai, X., et al. (2019). Ribs for enhanced flow distribution. *Applied Thermal Engineering*, 152, 343–352.
<https://doi.org/10.1016/j.applthermaleng.2019.02.108>
- [29] Li, P., et al. (2022). CFD optimization of multichannel heat sinks. *International Journal of Thermal Sciences*, 172, Article 107228.
<https://doi.org/10.1016/j.ijthermalsci.2021.107228>
- [30] Yadav, A., et al. (2023). Numerical investigation of baffled micro-channel heat sinks. *Applied Thermal Engineering*, 217, Article 119162.
<https://doi.org/10.1016/j.applthermaleng.2022.119162>
- [31] Al-Neama, M., et al. (2018). Multi-baffle heat sink for electronics cooling. *Entropy*, 20 (12), 979.
<https://doi.org/10.3390/e20120979>
- [32] Wu, T., et al. (2022). Laminar thermal management in compact heat sinks. *Energy Conversion and Management*, 260, Article 115633.
<https://doi.org/10.1016/j.enconman.2022.115633>
- [33] Chen, S., et al. (2024). Thermal uniformity enhancement in microchannel cooling. *International Journal of Heat and Mass Transfer*, 208, Article 124266.
<https://doi.org/10.1016/j.ijheatmasstransfer.2023.124266>
- [34] Park, H., et al. (2024). Thermal–hydraulic analysis of passive channel inserts. *Applied Thermal Engineering*, 235, Article 121434.
<https://doi.org/10.1016/j.applthermaleng.2023.121434>