



ARTICLE

Basic Study on Heat Transfer Enhancement in Straight Fins by Pulsating Flow

Tomoya Kazumi^{1,2}, Kota Fujisawa¹, Kenta Emori², Takashi Fukue^{1,*} and Yasuhiro Sugimoto¹

¹Department of Mechanical Engineering, Kanazawa Institute of Technology, Ishikawa, Japan

²Advanced Materials and Components Laboratory, Research Division, Nissan Motor Co., Ltd., Kanagawa, Japan

*Corresponding Author: Takashi Fukue. Email: fukue@neptune.kanazawa-it.ac.jp

Received: 09 July 2026; Accepted: 03 September 2026; Published: 27 October 2026

ABSTRACT: Efficient thermal management is essential for high-heat-flux electronic devices, where increasing power density and compact packaging impose strict requirements on cooling performance with minimal pressure drop. Pulsating flow has been proposed as a technique to enhance heat transfer; however, its effects on practical heat sink geometries remain insufficiently understood. This study investigates the thermal-hydraulic characteristics of pulsating flow in a multi-straight-fin heat sink operating in the laminar and transitional flow regimes ($Re = 480\text{--}2400$) through a combination of heat transfer experiments and three-dimensional Computational Fluid Dynamics (CFD) analyses. The effects of pulsation frequency, amplitude, and Reynolds number on heat-transfer and pressure-drop characteristics were evaluated using the mean Stanton number, friction coefficient, and performance evaluation criteria (PEC). The CFD model was validated against experimental measurements and used to analyze phase-resolved velocity and temperature fields. The results showed that pulsating flow can either enhance or deteriorate thermal performance depending on the pulsation conditions. Higher pulsation frequencies improved both the mean Stanton number and PEC, whereas lower frequencies reduced overall performance. The enhancement was most pronounced at an intermediate Reynolds number of approximately $Re = 960$. Increasing pulsation amplitude increased both heat transfer and pressure drop, resulting in little variation in PEC. Local heat-transfer evaluations demonstrated that the enhancement effect became more significant in the downstream region. CFD analyses revealed that periodic disturbances of the thermal boundary layer caused by unsteady near-wall flow were responsible for the enhanced heat transfer. These findings provide fundamental insights into the thermal-hydraulic characteristics and enhancement mechanisms of pulsating flow and support its application to compact liquid-cooled heat sinks for high-heat-flux electronic devices.

KEYWORDS: Heat transfer; pulsating flow; straight fin; CFD analysis; experiment

1 Introduction

The transition toward a carbon-neutral society has accelerated the development of next-generation mobility technologies and increased the widespread use of high-performance electronic devices across various fields. As these systems continue to advance, their increasing power density and compact packaging make efficient thermal management a critical requirement [1]. Power semiconductor devices, in particular, can generate more than 100 W of heat within a chip only a few millimeters in size due to conduction and switching losses [2]. Wide-bandgap semiconductors such as silicon carbide and gallium nitride further increase heat-flux levels, intensifying the demand for reliable and high-performance cooling solutions [3].

Liquid-cooled heat sinks are widely used for high-heat-flux electronics because they can dissipate large amounts of heat with relatively small temperature rises. Their performance depends on both heat-transfer

surface area and heat-transfer coefficient, and numerous studies have explored improvements through enhanced fin geometries, high-density structures, and turbulence-promoting features [4–6]. However, such geometric enhancements often increase flow resistance, leading to higher pressure drop and greater pumping-power requirements [7,8]. Therefore, alternative cooling approaches that enhance heat transfer without significantly increasing pressure drop are needed.

Pulsating flow, in which the coolant velocity is periodically modulated, has attracted increasing attention as a novel cooling technique. Pulsating flow exhibits periodic variations in velocity and pressure, similar to the pulsatile motion of blood driven by the heart. Studies on cardiovascular and artificial-heart systems have shown that introducing periodic fluctuations, compared with maintaining a constant flow rate, can reduce blood pressure and improve thermal regulation [9–11], suggesting its applicability as a biomimetic approach to enhancing heat transfer. Previous studies have reported that introducing pulsating flow into a rectangular channel or circular pipe can generate reverse flow during the deceleration phase, suppressing the development of the thermal boundary layer and thereby enhancing heat transfer [12–15]. Additional research has shown that applying pulsating flow around a cylinder enhances coolant supply to the separation region, thereby improving heat-transfer performance [16,17]. However, quantitative investigations of pulsating flow in practical straight-fin heat sinks remain limited, particularly in the laminar and transitional Reynolds number regimes that are widely encountered in electronic-cooling applications. Most previous studies have focused on simplified flow configurations such as circular pipes and rectangular channels, and the influence of pulsation parameters on heat-transfer characteristics in multi-fin heat sinks has not been fully clarified. Furthermore, the mechanisms responsible for heat-transfer enhancement under pulsating flow, including their relationship with the flow field, have not been sufficiently understood.

Against these backgrounds, this study investigates pulsating flow in a multi-straight-fin heat sink operating at Reynolds numbers between 480 and 2400, covering the laminar and transitional flow regimes. Heat transfer experiments and CFD analyses systematically evaluated how pulsation frequency, amplitude, and Reynolds number affect thermal-hydraulic performance. In addition, detailed flow-field analyses were performed to elucidate the mechanisms responsible for heat-transfer enhancement under pulsating flow. The findings provide fundamental insights into applying pulsating-flow cooling to compact liquid-cooled heat sinks for high-heat-flux electronic devices.

2 Experimental Methods

This section describes the heat transfer experiments conducted to evaluate the effects of pulsating flow on the straight-fin heat sink.

2.1 Test Heat Sink

Fig. 1a shows a schematic of the test heat sink. The outer dimensions of the rectangular channel are $W_o = 20$ mm in width and $H_o = 10$ mm in height, while the inner flow passage has dimensions $W_i = 16$ mm and $H_i = 6$ mm. The total length of the heat sink is $L = 3l$, consisting of an upstream developing section of length l , a finned section of length l , and a downstream section of length l , as illustrated in Fig. 1b. Fig. 1c shows a cross-sectional view of the finned channel in the y – z plane, which is used to illustrate the flow passage geometry and fin arrangement in the transverse direction. The finned section contained two straight fins, and each fin had the following dimensions. A height a was 6 mm, A width c was 2 mm, and a length l was 50 mm. The fin spacing b was 4 mm.

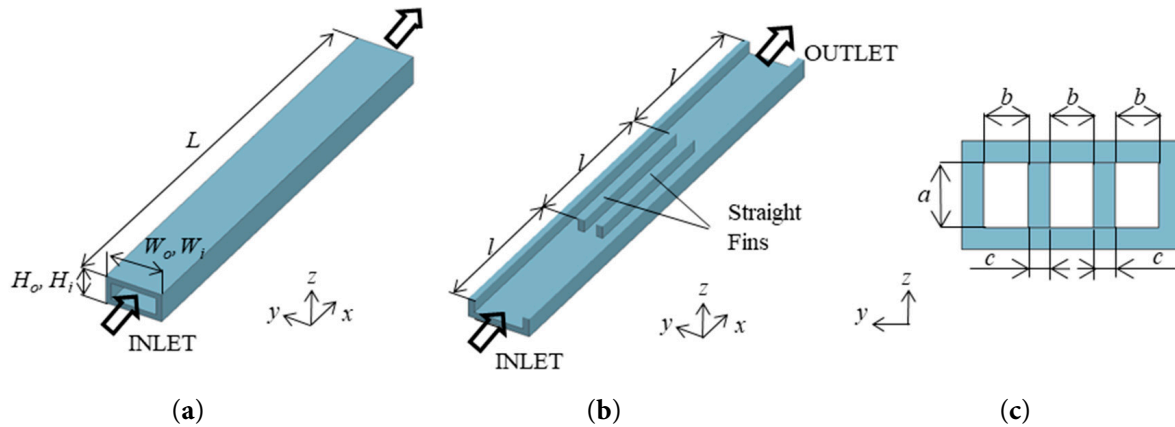


Figure 1: Geometry of the straight-fin heat sink. (a) Overall geometry of the rectangular channel, showing the outer dimensions $W_o = 20$ mm and $H_o = 10$ mm. (b) Schematic of the internal flow passage with upstream, finned, and downstream sections of equal length l . (c) Cross-sectional view of the finned channel in the y - z plane, showing two straight fins with height $a = 6$ mm, width $c = 2$ mm, spacing $b = 4$ mm.

2.2 Experimental System

Fig. 2 shows the details of the test section and the fabricated heat sink used in the experiments. Fig. 2a presents a photograph of the experimental heat sink, while Fig. 2b illustrates the fabricated heat-sink assembly, including the inlet and outlet flanges, threaded nipples, heater (SAKAGUCHI ELECTRIC HEATERS; WALN-9, Japan) controlled by a variable autotransformer (YAMABISHI ELECTRIC; S-130-10, Japan), and pressure-gauge mounting ports. A copper block was installed between the heater and the test section to promote uniform heat distribution and provide a nearly uniform heat flux to the heat sink. This fabricated model corresponds to the geometry described in Section 2.1. Fig. 2c illustrates the temperature-measurement location. Ten thermocouples (SAKAGUCHI ELECTRIC HEATERS; T-35, Japan) with an accuracy of $\pm 0.75\%$ of the measured temperature were inserted into 1-mm-diameter holes drilled from the bottom surface to the fin centerline at streamwise positions of $0.02l$, $0.26l$, $0.50l$, $0.74l$, and $0.98l$. Pressure ports were installed at the inlet and outlet to measure differential pressure.

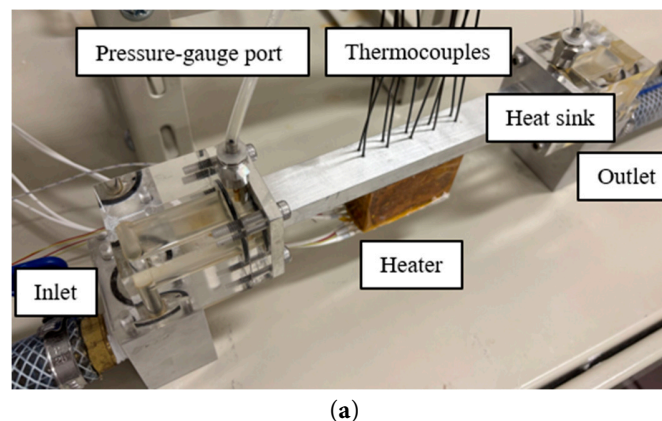


Figure 2: Cont.

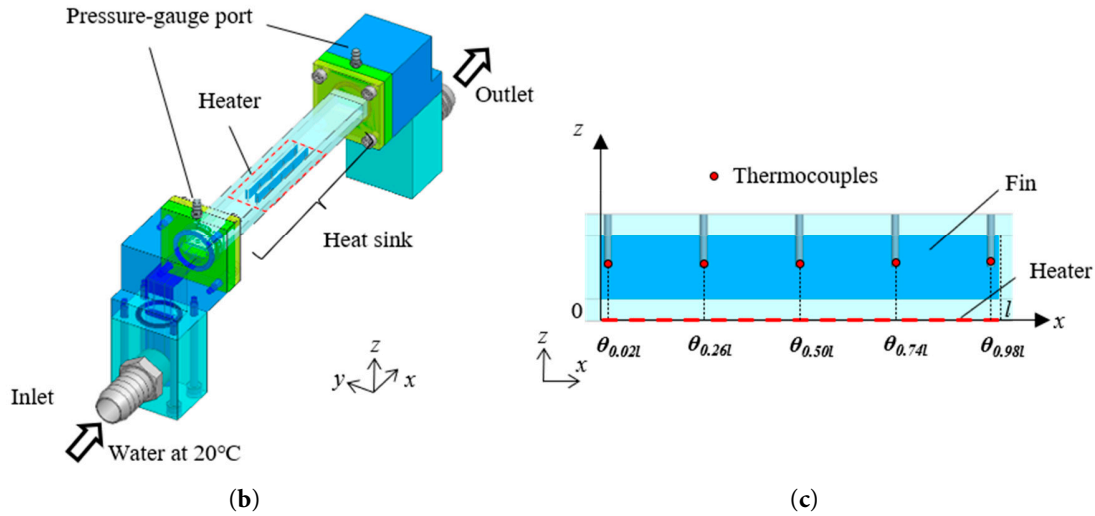


Figure 2: Setup of test section for thermal measurements; (a) Photograph of the experimental heat sink. (b) Detailed CAD model of the heat sink assembly including inlet/outlet flanges, threaded nipples, heater, and pressure-gauge ports. (c) Locations of thermocouples inserted at five streamwise positions ($0.02l$, $0.26l$, $0.50l$, $0.74l$, $0.98l$).

The whole experimental system used to generate the coolant flow is shown in Fig. 3. The system consists of a chiller (SMC; HRS018-A-10, Japan), a manual valve, a proportional control valve (KOGANEI; KFPV300-80-FM-S13-04, Japan), a flow meter (KEYENCE; FD-SS20A, Japan) with an accuracy of $\pm 4\%$ of the reading, differential pressure gauge (BD sensors; DPT100, Germany) with an accuracy of $\pm 0.1\%$ F.S., the heatsink mentioned in Section 2.1, and the return line to the chiller, arranged in this order from upstream to downstream [18]. The chiller supplies a constant flow rate of 20°C water, which is first adjusted using the manual valve. The proportional control valve opening then generates steady and pulsating flows using a dedicated controller (KOGANEI; KFPC1-F07-DN, Japan). A microcontroller (Arduino; Arduino UNO, Italy) generates the control signal and outputs a sinusoidal waveform to modulate the valve opening. To ensure that the flow velocity waveform, temperature, and pressure had reached quasi-steady conditions, measurements were conducted for 1 min starting 5 min after the beginning of each experiment.

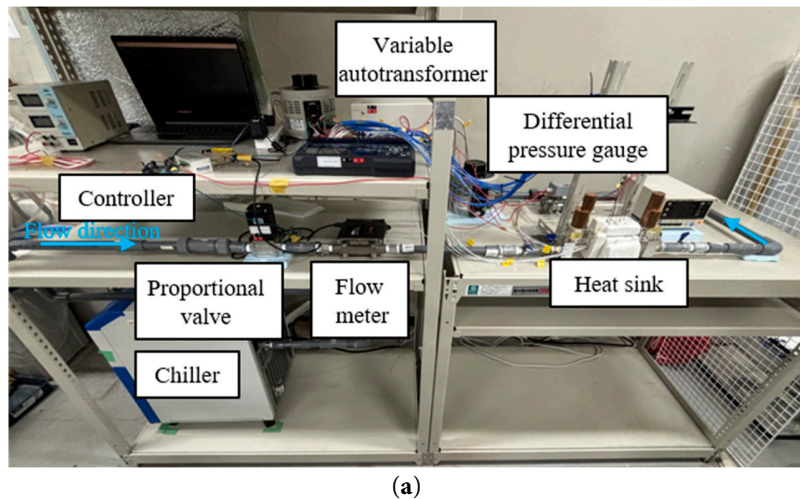


Figure 3: Cont.

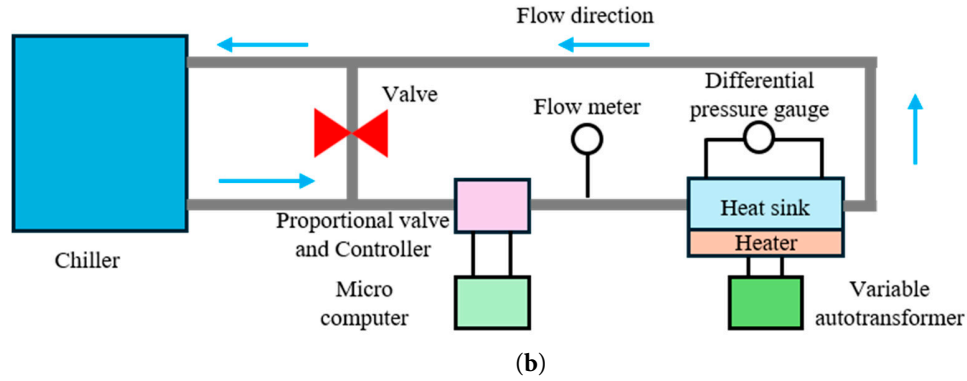


Figure 3: Experimental system for generating steady and pulsating flow; (a) Photograph of the experimental apparatus. (b) Schematic diagram of the experimental system.

2.3 Waveform of Pulsating Flow

The target velocity waveforms of the steady and pulsating flows supplied to the inlet of the heat sink are expressed by Eq. (1), and the corresponding waveform parameters are summarized in Table 1. The pulsating flow is represented by a sinusoidal waveform with three parameters: frequency f , amplitude A , and time-averaged velocity V_m . Representative velocity waveforms are shown in Fig. 4. First, Fig. 4a corresponds to a steady flow condition with frequency $f = 0$ Hz, amplitude $A = 0$, and time-averaged velocity $V_m = 0.2$ m/s. In contrast, Fig. 4b corresponds to a pulsating flow condition with frequency $f = 1.0$ Hz, amplitude $A = 1.0$, and time-averaged velocity $V_m = 0.2$ m/s. Fig. 4c shows a case with a smaller amplitude and higher frequency, namely frequency $f = 2.0$ Hz, amplitude $A = 0.5$, and time-averaged velocity $V_m = 0.2$ m/s. These waveforms are used as typical examples to illustrate how waveform parameters influence the inlet velocity profile.

$$V_t = (1 + A \sin 2\pi ft)V_m \quad (1)$$

Table 1: Parameters defining the steady and pulsating inlet velocity waveforms.

Parameter	Unit	Value
f , Frequency	Hz	0, 0.5, 1.0, 2.0, 3.0, 4.0
A , Amplitude	-	0, 0.25, 0.50, 1.00
V_m , Time-averaged velocity	m/s	0.1, 0.2, 0.3, 0.4, 0.5

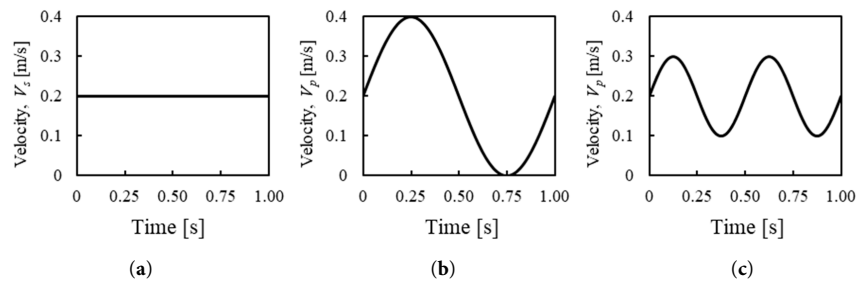


Figure 4: Representative target velocity waveforms for steady and pulsating flow; (a) Steady flow with frequency $f = 0$ Hz, amplitude $A = 0$ and time-averaged velocity $V_m = 0.2$ m/s. (b) Pulsating flow with frequency $f = 1.0$ Hz, amplitude $A = 1.0$ and time-averaged velocity $V_m = 0.2$ m/s. (c) Pulsating flow with frequency $f = 2.0$ Hz, amplitude $A = 0.5$ and time-averaged velocity $V_m = 0.2$ m/s.

3 CFD Analysis

The CFD analysis was conducted to verify the reliability and consistency of the experimental results. A three-dimensional CFD model of the heat sink was constructed based on the geometry described in Section 2.1, and analyses were performed under the same inlet conditions as the experiments.

3.1 Governing Equations

The flow and heat-transfer fields were obtained by solving the continuity equation in Eq. (2), the incompressible Navier–Stokes equations in Eq. (3), and the energy equation in Eq. (4). The fluid was assumed to be incompressible, and the thermophysical properties of water at 20°C were used in all analyses.

$$\nabla \cdot \vec{V} = 0 \quad (2)$$

$$\rho \left(\frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V} \right) = -\nabla p + \mu \nabla^2 \vec{V} \quad (3)$$

$$\rho C \left(\frac{\partial \theta}{\partial t} + \vec{V} \cdot \nabla \theta \right) = \nabla \cdot (\lambda \nabla \theta) \quad (4)$$

3.2 Computational Domain and Meshes

The CFD Analysis was conducted using the commercial CFD solver STAR-CCM+ (Siemens Digital Industries Software, Ver. 2410). The computational domain consisted of the upstream developing section, the finned section, and the downstream section, with a total length of $L = 3l$.

A polyhedral mesh was employed for the fluid region, and prism-layer meshes were applied at the solid–fluid interfaces to accurately capture near-wall gradients. An overview of the mesh is shown in Fig. 5a, and an enlarged view near the fin–fluid interface is presented in Fig. 5b. The detailed mesh conditions used in the analysis are summarized in Table 2. We conducted a grid-independence study by varying the time step and mesh size, resulting in Courant numbers ranging from 100 to 0.5. The results showed that the predicted Stanton number converged when the Courant number was below 10. Balancing accuracy and cost, we adopted a Courant number of 1.0. At this Courant number, the simulation used an inlet velocity of 0.5 m/s, a base mesh size of 5.0×10^{-4} m, and a time step of 1.0×10^{-3} s, resulting in a computational mesh of approximately 10 million cells.

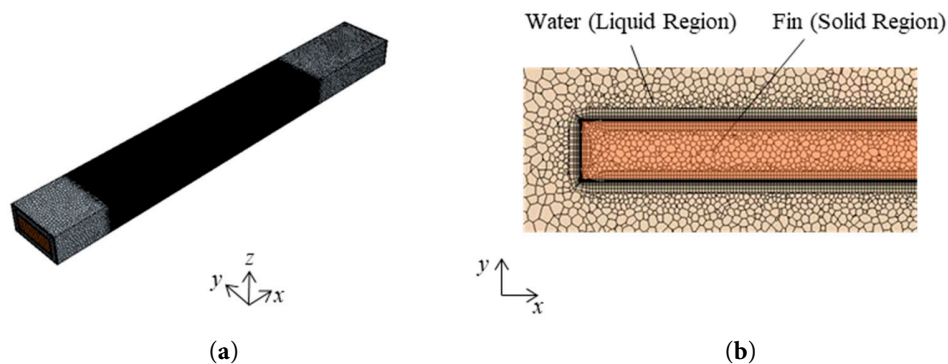


Figure 5: Computational meshes used in the CFD analysis; (a) Overview of the polyhedral mesh for the entire computational domain. (b) Enlarged view of the prism-layer mesh near the fin–fluid interface.

Table 2: Mesh conditions used in the CFD analysis.

Mesh Type	Polyhedral Mesh
Near-wall Treatment	Prism-layer mesh
Total number of cells	Approx. 10,000,000
Base mesh size	5.0×10^{-4} m
Time step	1.0×10^{-3} s
Courant number	<1.0

3.3 Boundary and Initial Conditions

The inlet velocity was prescribed according to the steady or sinusoidal pulsating waveforms defined in Eq. (1) and Table 1. The outlet boundary was set to a zero-gauge-pressure condition. A uniform heat flux corresponding to the experimental heater input was applied to the heated surface. All other walls were treated as no-slip. At the start of the simulation, the fluid velocity and heat input were set to zero. Subsequently, the prescribed inlet flow and heat flux were imposed, and the temperature distribution was extracted after 20 s, when the solution was confirmed to be in a quasi-steady state.

4 Evaluation Methods

4.1 Reynolds Number

The cooling performance under steady and pulsating flow conditions was evaluated using the Stanton number, friction coefficient, and performance evaluation criteria (PEC) [19–22]. In this research, the Reynolds number used in this study was defined in Eq. (5) using the mean flow velocity V_m [m/s], the hydraulic diameter D [m] defined in Eq. (6), and the kinematic viscosity of water at 20°C ν [m²/s]. Based on the mean flow velocities listed in Table 1, the Reynolds number investigated in this study ranged from 480 to 2400.

$$Re_m = \frac{V_m D}{\nu} \quad (5)$$

$$D = \frac{2ab}{a+b} \quad (6)$$

4.2 Stanton Number

The Stanton number was calculated to evaluate the cooling performance of the heat sink under steady and pulsating flow conditions. First, the instantaneous local heat-transfer coefficient at each temperature-measurement position was calculated using Eq. (7), where θ_w [K] is the bulk water temperature at 20°C maintained by the chiller and S [m²] is the wetted surface area defined in Eq. (8). In Eq. (8), the first term represents the side surface area of the fins, while the second term corresponds to the frontal and rear surface areas of the fins.

The instantaneous local Stanton number was then obtained using Eq. (9), where ρ [kg/m³] is the coolant density and c_p [J/(kg·K)] is the specific heat of the coolant. The time-averaged local Stanton number over one pulsation cycle was calculated using Eq. (10), where T [s] denotes the duration of one pulsation cycle. Next, the time-averaged local Stanton numbers at the five streamwise positions (0.02*l*, 0.26*l*, 0.50*l*, 0.74*l*, and 0.98*l*) were averaged using Eq. (11).

$$h_{x,t} = \frac{Q}{S(\theta_{x,t} - \theta_w)} \quad (7)$$

$$S = 6l(a + b) + 4ac \quad (8)$$

$$St_{x,t} = \frac{h_{x,t}}{\rho c_p V_m} \quad (9)$$

$$St_x = \frac{1}{T} \int_0^T St_{x,t} dt \quad (10)$$

$$St_m = \frac{1}{l} \int_0^l St_x dx = \frac{St_{0.02l} + St_{0.26l} + St_{0.50l} + St_{0.74l} + St_{0.98l}}{5} \quad (11)$$

4.3 Friction Coefficient

The friction coefficient was calculated to evaluate the pressure drop characteristics of the heat sink under steady and pulsating flow conditions. First, the instantaneous pressure difference between the inlet and outlet of the heat sink was obtained using Eq. (12). The time-averaged pressure drops over one pulsation cycle were then calculated using Eq. (13), where T [s] denotes the duration of one pulsation cycle. The mean friction coefficient of the fins was subsequently determined using Eq. (14), based on the time-averaged pressure drop, the mean flow velocity, and the hydraulic diameter of the flow passage.

$$\Delta P_t = P_{in,t} - P_{out,t} \quad (12)$$

$$\Delta P_m = \frac{1}{T} \int_0^T \Delta P_t dt \quad (13)$$

$$Cf_m = \frac{2D\Delta P_m}{\rho l V_m^2} \quad (14)$$

4.4 Performance Evaluation Criteria

The PEC was calculated to evaluate the overall thermal-hydraulic performance of the heat sink by considering the trade-off between heat-transfer enhancement and the accompanying increase in pressure drop [23]. The PEC is commonly used for assessing heat exchangers and cooling devices and was obtained using Eq. (15), which incorporates both the mean Stanton number and the mean friction coefficient. The PEC provides a comprehensive indicator of cooling performance by simultaneously accounting for heat-transfer improvement and flow resistance, enabling a balanced comparison between steady and pulsating flow conditions.

$$\eta = \frac{St_m}{Cf_m^{\frac{1}{3}}} \quad (15)$$

5 Experimental and Analytical Results

5.1 Comparison of Velocity Waveforms in Experiments and CFD Analyses

Fig. 6 compares the target velocity waveforms defined in Eq. (1) with the measured waveforms obtained in the experiments and the inlet velocity imposed in the CFD analysis. In the experiments, the actual velocity waveforms deviated slightly from the ideal target waveforms due to the dynamic response of the proportional control valve and the flow loop. In particular, the experimentally measured waveforms exhibited relatively steeper acceleration and deceleration phases than the ideal target waveforms. Nevertheless, the measured waveforms closely followed the intended profiles, and the deviations were sufficiently small so that the target waveforms were used as the inlet boundary condition in the CFD analysis. The simulated waveforms reproduced the target profiles with high fidelity, confirming the consistency between the experimental and CFD inlet conditions.

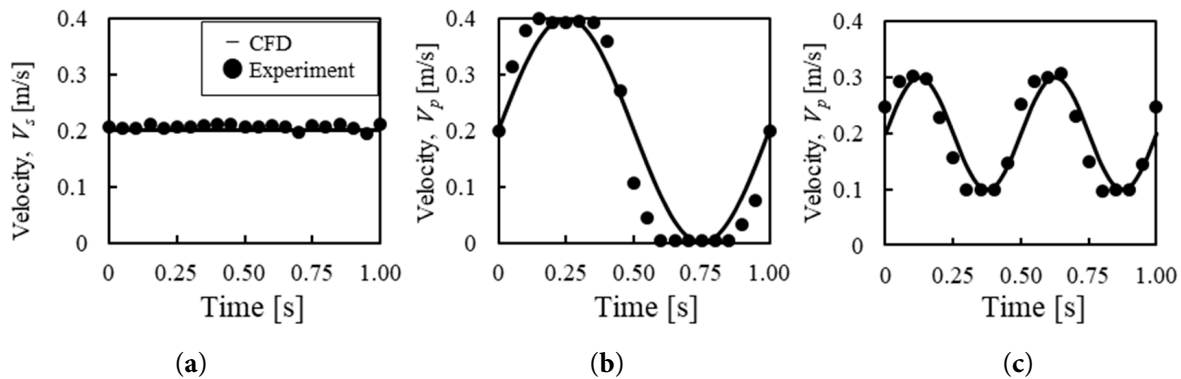


Figure 6: Comparison of inlet velocity waveforms in CFD analysis and experiments; (a) Steady flow with frequency $f = 0$ Hz, amplitude $A = 0$ and time-averaged velocity $V_m = 0.2$ m/s. (b) Pulsating flow with frequency $f = 1.0$ Hz, amplitude $A = 1.0$ and time-averaged velocity $V_m = 0.2$ m/s. (c) Pulsating flow with frequency $f = 2.0$ Hz, amplitude $A = 0.5$ and time-averaged velocity $V_m = 0.2$ m/s.

5.2 Evaluation Results in Experiments and CFD Analyses

The cooling performance under steady and pulsating flow conditions was evaluated using the mean Stanton number, friction coefficient, and performance evaluation criteria (PEC). This section presents the effects of pulsation frequency, pulsation amplitude, and Reynolds number on the thermal–hydraulic performance of the heat sink, based on both experimental measurements and CFD analyses.

5.2.1 Effect of Pulsation Frequency

Fig. 7 shows the variations in the mean Stanton number, friction coefficient, and PEC as functions of pulsation frequency. At low frequencies (e.g., $f = 0.5$ Hz), the mean Stanton number decreased relative to steady flow. As the pulsation frequency increased, the mean Stanton number increased and exceeded the steady-flow value. The friction coefficient increased at low frequencies and gradually decreased with increasing frequency, although it remained higher than that of steady flow. As a result, the PEC decreased at low frequencies and increased at higher frequencies. Both the experimental and CFD results exhibited these same trends, demonstrating that the CFD model accurately reproduces the frequency dependence of cooling performance.

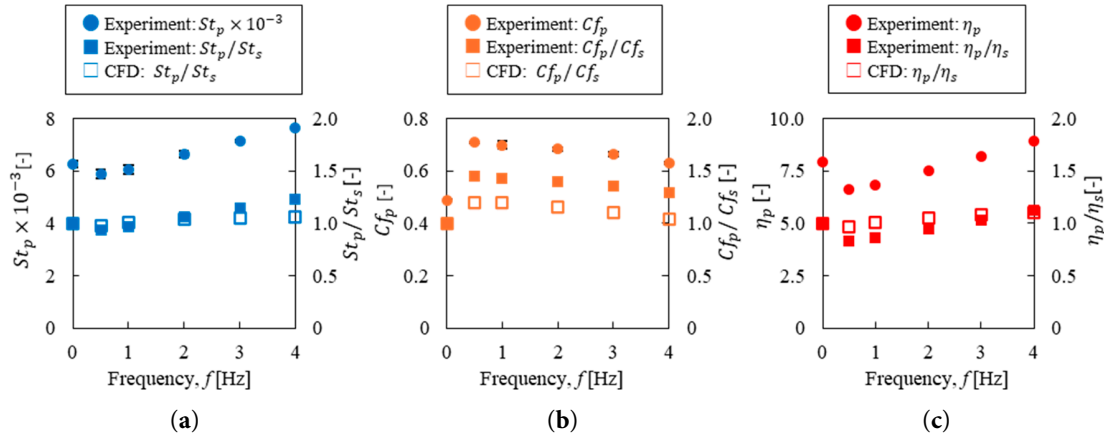


Figure 7: Effects of pulsation frequency on heat transfer and pressure drop characteristics with amplitude $A = 1.0$ and time-averaged velocity $V_m = 0.2$ m/s; (a) Stanton number. (b) Friction coefficient. (c) Performance evaluation criteria (PEC). The experimental results in (a,b) are presented as mean values $\pm$ standard deviation.

5.2.2 Effect of Pulsation Amplitude

Fig. 8 shows that the mean Stanton number increased with increasing pulsation amplitude. The friction factor also increased due to the larger velocity fluctuations. As a result, the PEC remained nearly constant across the amplitude range investigated. The CFD analysis captured the same monotonic increase in Stanton number and friction factor, with only minor quantitative differences, indicating that the CFD model reliably predicts amplitude-dependent cooling performance.

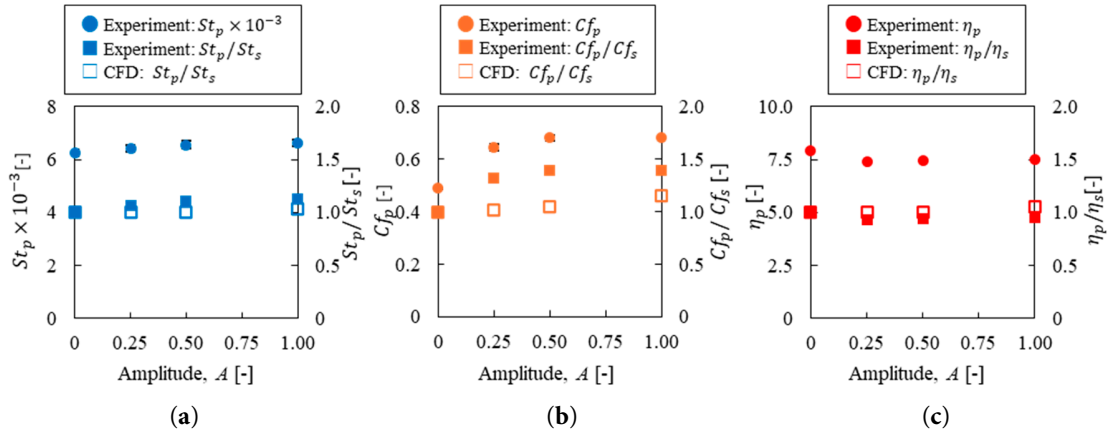


Figure 8: Effects of pulsation amplitude on heat transfer and pressure drop characteristics with frequency $f = 2.0$ Hz and time-averaged velocity $V_m = 0.2$ m/s; (a) Stanton number. (b) Friction coefficient. (c) Performance evaluation criteria (PEC). The experimental results in (a,b) are presented as mean values $\pm$ standard deviation.

5.2.3 Effect of Reynolds Number

Fig. 9 shows the influence of the Reynolds number. The enhancement of the mean Stanton number due to pulsation was most pronounced at $Re = 960$. At higher Reynolds numbers, although the absolute Stanton number increased, the relative enhancement due to pulsation decreased. The friction coefficient increased with Reynolds number under pulsating flow, and the PEC exhibited its maximum improvement

at $Re = 960$. The CFD results reproduced these Reynolds-number-dependent trends, further confirming the consistency between the experimental and computational approaches.

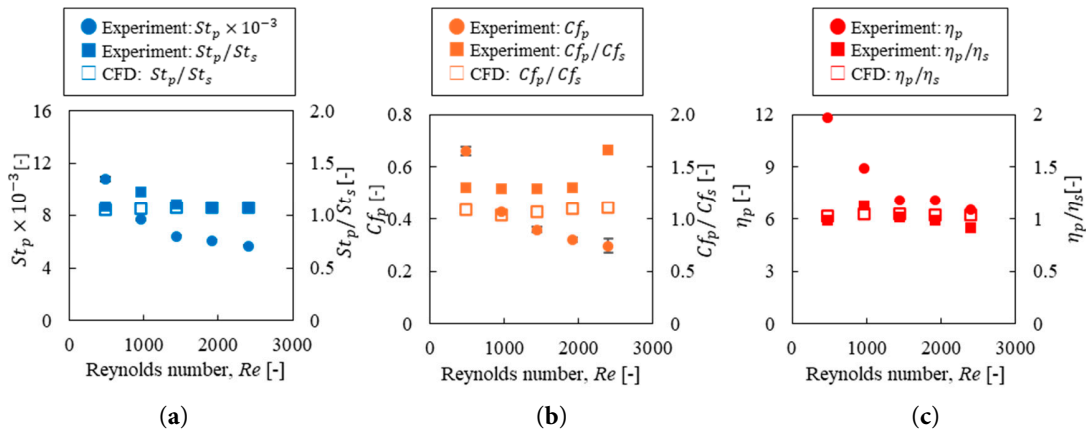


Figure 9: Effects of Reynolds number on heat transfer and pressure drop characteristics with frequency $f = 4.0$ Hz and amplitude $A = 1.0$; (a) Stanton number. (b) Friction coefficient. (c) Performance evaluation criteria (PEC). The experimental results in (a,b) are presented as mean values $\pm$ standard deviation.

5.2.4 Local Temperature and Stanton Number Distribution

This section focuses on a detailed examination of the cooling performance enhancement induced by pulsating flow. Fig. 10 shows the local Stanton number ratio between pulsating and steady flows, together with the local fin temperature, evaluated at five streamwise locations from the upstream to downstream region of the fin for the condition of $A = 1.0$ and $V_m = 0.2$ m/s. The local temperature distribution under steady-flow conditions indicates that the fin temperature is lower in the upstream region and generally increases toward the downstream region. This trend is attributed to the development of the thermal boundary layer along the fin surface, which reduces the local heat-transfer coefficient downstream. However, the temperature at $x = 0.98l$ is lower than that at $x = 0.74l$. This is attributed to additional heat dissipation from the heat-sink housing near the downstream end, which suppresses the local temperature rise. The local Stanton number ratio demonstrates a clear dependence on streamwise position. Under pulsating-flow conditions, the enhancement of the Stanton number becomes more pronounced toward the downstream region of the fin. In contrast, the increase in the upstream region is relatively small. These results suggest that pulsating flow is particularly effective in the downstream region, where the thermal boundary layer is more fully developed. The CFD results capture the same streamwise trend as the experimental data, supporting the observed distribution of local heat-transfer enhancement.

6 Visualization of Flow Velocity and Temperature Distribution from CFD Analysis

To clarify why pulsating flow enhances cooling performance, particularly why the enhancement becomes more pronounced in the downstream region of the fin, as observed in Section 5, the instantaneous velocity and temperature fields obtained from CFD analyses were analyzed under both steady and pulsating-flow conditions. For the pulsating-flow case, contour plots were extracted at four representative phases within one pulsation period to examine the temporal evolution of the velocity and thermal boundary layers.

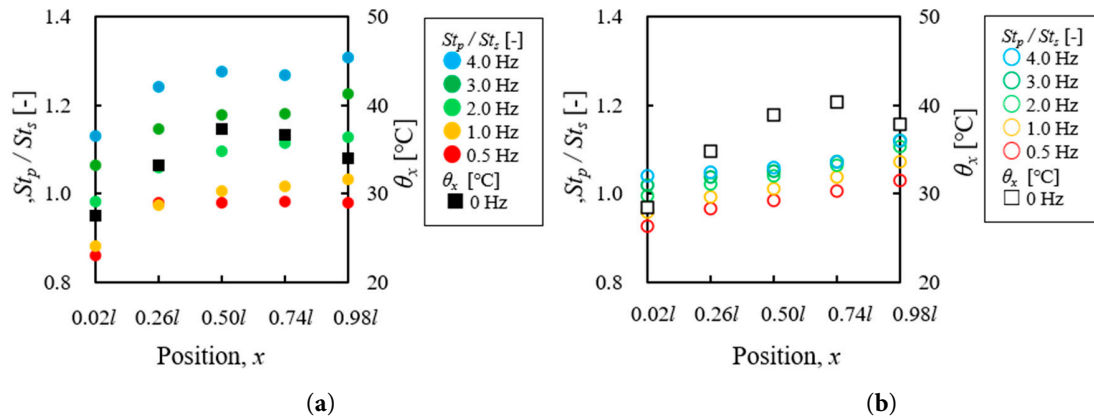


Figure 10: Local Stanton number ratio at five streamwise locations from upstream to downstream along the fin under the condition of $A = 1.0$ and $V_m = 0.2$ m/s; (a) experimental results and (b) Analysis results.

6.1 Upstream of the Fins

In the upstream region of the fin, the thermal boundary layer under steady flow is inherently thin, as confirmed by the temperature contours shown in Fig. 11a. Owing to this thin boundary layer, the local heat-transfer performance is already high, leaving limited potential for enhancement by flow pulsation. Under pulsating flow, phase-dependent variations are observed in both the velocity and temperature fields. As the flow decelerates from $t = 1/4T$ to $t = 3/4T$, the local velocity decreases, and the thermal boundary layer begins to show disturbances, as indicated in Fig. 11a. At $t = 3/4T$, the velocity contours in Fig. 11b reveal reverse flow near the fin surface, which transports high-temperature fluid adjacent to the fin surface toward the center region between the fins. At $t = T$, this high-temperature fluid is subsequently discharged downstream.

However, because the thermal boundary layer is not well developed in the upstream region even under steady flow conditions, the influence of pulsating flow on the local thermal field remains limited compared with that in the downstream region.

6.2 Downstream of the Fins

In contrast, the downstream region of the fin exhibits a significantly thicker thermal boundary layer under steady flow, as shown by the temperature contours in Fig. 12a. The development of this thermal boundary layer leads to reduced local heat-transfer performance and provides greater potential for enhancement by pulsating flow. Similar phase-dependent behavior is observed under pulsating flow. As the flow decelerates from $t = 1/4T$ to $t = 3/4T$, disturbances in the thermal boundary layer become more pronounced in the downstream region, as seen in Fig. 12a.

At $t = 3/4T$, the velocity contours in Fig. 12b indicate the occurrence of reverse flow near the fin surface, which transports high-temperature fluid from the fin surface toward the center of the fin channel. This transport is particularly prominent near the fin's trailing edge. At $t = T$, the accumulated high-temperature fluid is discharged downstream.

Because the thermal boundary layer is well developed in the downstream region, the effect of pulsating flow on the local thermal field is more pronounced than in the upstream region, resulting in a greater enhancement in cooling performance.

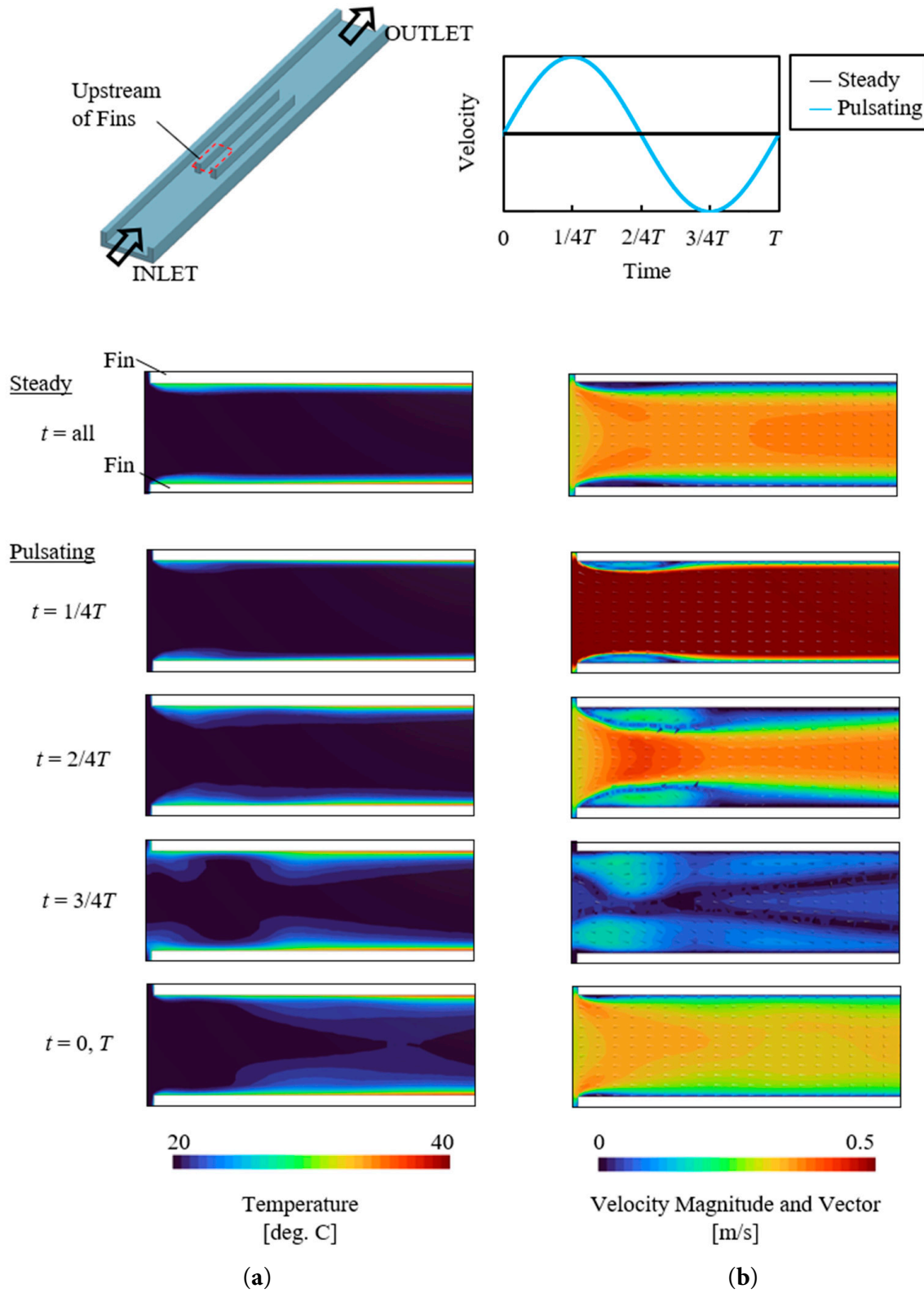


Figure 11: Visualizations of velocity and temperature distributions at the upstream region of the fins under steady flow ($V_m = 0.2 \text{ m/s}$) and pulsating flow ($A = 1.0$, $f = 2.0 \text{ Hz}$ and $V_m = 0.2 \text{ m/s}$) conditions; **(a)** Temperature distributions, **(b)** Velocity distributions.

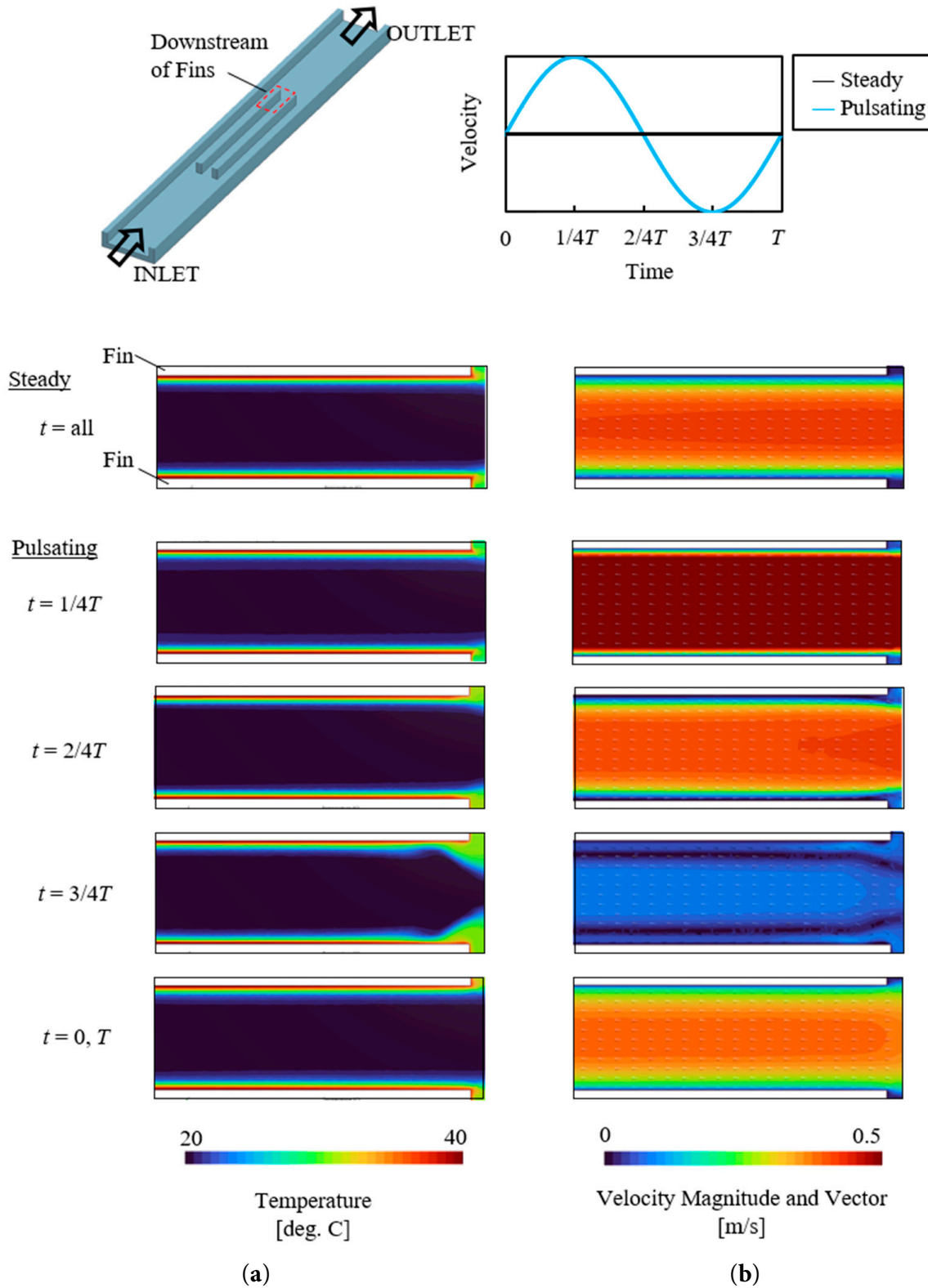


Figure 12: Visualizations of velocity and temperature distributions at the downstream region of the fins under steady flow ($V_m = 0.2$ m/s) and pulsating flow ($A = 1.0$, $f = 2.0$ Hz and $V_m = 0.2$ m/s) conditions; (a) Temperature distributions. (b) Velocity distributions.

7 Conclusion

This study experimentally and analytically investigated the thermal–hydraulic performance of a multi-straight-fin heat sink operating in the laminar and transitional flow regimes ($Re = 480–2400$) under steady and sinusoidally pulsating flow conditions. The effects of pulsation frequency, pulsation amplitude, and Reynolds number were evaluated using the mean Stanton number, friction coefficient, and performance evaluation criteria (PEC). The experimental and CFD results showed good qualitative agreement over the entire range of conditions.

The results show that pulsating flow can either enhance or degrade cooling performance depending on the pulsation parameters. At low pulsation frequencies, the mean Stanton number decreased, whereas the friction coefficient increased relative to steady flow, resulting in a reduction in PEC. In contrast, increasing the pulsation frequency resulted in a higher mean Stanton number than that of steady flow and an accompanying increase in PEC. The enhancement was most pronounced at an intermediate Reynolds number of $Re = 960$. Increasing the pulsation amplitude increased both the heat-transfer rate and pressure drop, yielding an approximately constant PEC within the investigated range.

Evaluation of local Stanton number distributions demonstrated that heat-transfer enhancement by pulsating flow is spatially non-uniform along the fin length. At lower frequencies, the enhancement was more prominent in the downstream region, whereas higher-frequency pulsation enhanced heat transfer over a wider streamwise range. Phase-resolved CFD analyses showed that in the downstream region, where the thermal boundary layer is well developed under steady flow, pulsating flow introduces periodic variations in the near-wall velocity and temperature fields. These unsteady effects intermittently disturb the thermal boundary layer, resulting in greater relative enhancement than in the upstream region.

Overall, this study systematically clarified the effects of pulsation frequency, amplitude, and Reynolds number on the thermal-hydraulic performance of a multi-straight-fin heat sink in the laminar-to-transition Reynolds number regime. Furthermore, the combined experimental and CFD investigation elucidated the flow-field mechanisms responsible for heat-transfer enhancement under pulsating flow. The findings provide fundamental insights into the thermal-hydraulic characteristics and heat-transfer enhancement mechanisms of pulsating flow in compact liquid-cooled heat sinks and contribute to the development of high-performance thermal-management systems for high-heat-flux electronic devices. Future work will extend the investigation to different fin geometries, working fluids, pulsation waveforms and heat sink design considering conjugate heat transfer and fin efficiency to further assess the applicability of pulsating flow to advanced thermal-management systems.

Acknowledgement: The authors would like to thank the members of the Fukue Laboratory, Kanazawa Institute of Technology, and the members of the Advanced Materials and Components Laboratory, Nissan Motor Co., Ltd., for their valuable technical support and helpful discussions.

Funding Statement: The authors received no specific funding for this study.

Author Contributions: The authors confirm contributions to the paper as follows: conceptualization: Tomoya Kazumi, Kenta Emori, Takashi Fukue and Yasuhiro Sugimoto; experimental investigation: Tomoya Kazumi and Kota Fujisawa; numerical model development: Tomoya Kazumi, Kota Fujisawa and Kenta Emori; numerical model validation: Takashi Fukue and Yasuhiro Sugimoto; draft manuscript preparation: Tomoya Kazumi, Kota Fujisawa and Kenta Emori; manuscript review: Takashi Fukue and Yasuhiro Sugimoto. All authors reviewed and approved the final version of the manuscript.

Availability of Data and Materials: The data and materials used in this study are available from the authors upon reasonable requests, subject to company confidentiality restrictions.

Ethics Approval: Not applicable.

Conflicts of Interest: The authors declare no conflicts of interest.

Nomenclature

Q	Heat input [W]
P	Static pressure [Pa]
V	Velocity [m/s]
A	Amplitude [-]
f	Frequency [Hz]
T	Period [s]
t	Time [s]
S	Area [m ²]
θ	Temperature [K]
h	Heat transfer coefficient [W/(m ² ·K)]
St	Stanton number [-]
c _p	Specific heat under constant pressure condition [J/(kg·K)]
C _f	Friction coefficient [-]
η	Performance evaluation criteria [-]
ν	Kinematic viscosity [m ² /s]

Subscript

s	Steady flow
p	Pulsating flow
in	Inlet
out	Outlet
t	Instantaneous value
x	Local value
m	Average
w	Water

References

1. Abramushkina E, Zhaksylyk A, Geury T, El Baghdadi M, Hegazy O. A thorough review of cooling concepts and thermal management techniques for automotive WBG inverters: topology, technology and integration level. *Energies*. 2021;14(16):4981. [[CrossRef](#)].
2. Joshi SN, Zhou F, Liu Y, Lohan DJ, Ukegawa H, Lee J, et al. A review of select patented technologies for cooling of high heat flux power semiconductor devices. *IEEE Trans Power Electron*. 2023;38(6):6790–4. [[CrossRef](#)].
3. Keller J, Brooks A. Impact of wide bandgap semiconductors (SiC/GaN) on next-generation EV power electronics. *Int J Comput Intell Syst*. 2025:1–13.
4. Gaikwad A, Sathe A, Sanap S. A design approach for thermal enhancement in heat sinks using different types of fins: a review. *Front Therm Eng*. 2023;2:980985. [[CrossRef](#)].
5. Alimehr A, Nikniazi A, Jalali A, Daymond MR, Pahlevani M, Fallah V. Heat sink design investigation via CFD thermal analysis and additive manufacturing rapid prototyping. *Int J Adv Manuf Technol*. 2025;141(7):4473–87. [[CrossRef](#)].
6. Rahman MA, Hasnain SMM, Paramasivam P, Ayanie AG. Advancing thermal management in electronics: a review of innovative heat sink designs and optimization techniques. *RSC Adv*. 2024;14(43):31291–319. [[CrossRef](#)].
7. Muhammad A, Selvakumar D, Iranzo A, Sultan Q, Wu J. Comparison of pressure drop and heat transfer performance for liquid metal cooled mini-channel with different coolants and heat sink materials. *J Therm Anal Calorim*. 2020;141(1):289–300. [[CrossRef](#)].
8. Zhou W, Dong K, Sun Q, Luo W, Zhang B, Guan S, et al. Research progress of the liquid cold plate cooling technology for server electronic chips: a review. *Int J Energy Res*. 2022;46(9):11574–95. [[CrossRef](#)].

9. Ozturk C, Rosalia L, Roche ET. A multi-domain simulation study of a pulsatile-flow pump device for heart failure with preserved ejection fraction. *Front Physiol.* 2022;13:815787. [[CrossRef](#)].
10. Bierewirtz T, Narayanaswamy K, Giuffrida R, Rese T, Bortis D, Zimpfer D, et al. A novel pumping principle for a total artificial heart. *IEEE Trans Biomed Eng.* 2024;71(2):446–55. [[CrossRef](#)].
11. Deutsch S, Tarbell JM, Manning KB, Rosenberg G, Fontaine AA. Experimental fluid mechanics of pulsatile artificial blood pumps. *Annu Rev Fluid Mech.* 2006;38:65–86. [[CrossRef](#)].
12. Blythman R, Alimohammadi S, Jeffers N, Murray DB, Persoons T. Heat transfer analysis of laminar pulsating flow in a rectangular channel using infrared thermography. *J Heat Transf.* 2022;144:011801. [[CrossRef](#)].
13. Blythman R, Persoons T, Jeffers N, Murray DB. Heat transfer of laminar pulsating flow in a rectangular channel. *Int J Heat Mass Transf.* 2019;128:279–89. [[CrossRef](#)].
14. Kumavat PS, Blythman R, Murray DB, O'Shaughnessy SM. Study of heat transfer enhancement by pulsating flow in a rectangular mini channel. In: *Advances in heat transfer and thermal engineering*. Singapore, Singapore: Springer; 2021. p. 253–7. [[CrossRef](#)].
15. Ren Y, Guo Z, Hu ZC. Numerical study of pulsating pipe flows and enhanced heat transfer of supercritical CO₂. *Int Commun Heat Mass Transf.* 2025;167:109124. [[CrossRef](#)].
16. Li G, Zheng Y, Yang H, Xu Y. Numerical investigation of heat transfer and fluid flow around the rectangular flat plane confined by a cylinder under pulsating flow. *J Appl Fluid Mech.* 2016;9(6):1569–77. [[CrossRef](#)].
17. El-Salamony M, Aziz M, Benini E, Gaheen O. Optimization study of unsteady flow affecting on cooling a heated cylinder. In: *Proceedings of the 2021 3rd Novel Intelligent and Leading Emerging Sciences Conference (NILES)*; 2021 Oct 23–25; Giza, Egypt. p. 175–8. [[CrossRef](#)].
18. Zhang D, Wu Y, Zhu K, Shi R, Ruan J, Ding C. Study of high-frequency pulsating flow control valves. *Appl Sci.* 2024;14(22):10177. [[CrossRef](#)].
19. Fukue T, Hiratsuka W, Shirakawa H, Nakayama K, Koito Y. Cfd-based study on heat transfer enhancement behind a projection in a miniaturized flow channel by pulsating flow. *Front Heat Mass Transf.* 2020;15:1–8. [[CrossRef](#)].
20. Hayakawa S, Fukue T, Sugimoto Y, Hiratsuka W, Shirakawa H, Koito Y. Effect of rib height on heat transfer enhancement by combination of a rib and pulsating flow. *Front Heat Mass Transf.* 2022;18(1):1–9. [[CrossRef](#)].
21. Ye Q, Zhang Y, Wei J. A comprehensive review of pulsating flow on heat transfer enhancement. *Appl Therm Eng.* 2021;196:117275. [[CrossRef](#)].
22. Abdelghany MT, Elshamy SM, Sharafeldin MA, Abdellatif OE. Experimental investigation on the effect of pulsating flow on heat transfer and pressure drop in conical tubes. *J Therm Anal Calorim.* 2023;148(13):6169–82. [[CrossRef](#)].
23. Webb RL, Eckert ERG. Application of rough surfaces to heat exchanger design. *Int J Heat Mass Transf.* 1972;15(9):1647–58. [[CrossRef](#)].