Convective heat transfer performance of airfoil heat sinks fabricated by selective laser melting

J.Y. Ho a, K.K. Wong a, K.C. Leong a*, T.N. Wong b

a Singapore Centre for 3D Printing, School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Republic of Singapore

b School of Mechanical and Aerospace Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore 639798, Republic of Singapore

ABSTRACT

This paper presents the forced convective heat transfer performances of novel airfoil heat sinks produced by Selective Laser Melting (SLM). Heat sinks with staggered arrays of NACA 0024 and NACA 4424 airfoil shaped fins were investigated experimentally and the results were compared with conventional heat sinks with circular and rounded rectangular fins. In addition, NACA 0024 heat sinks with angles of attack \( \alpha \) ranging from 0° to 20° were also fabricated and the effects of the angle of attack \( \alpha \) on the heat sink thermal performances were examined. Experiments were conducted in a rectangular air flow channel with tip \( (CL_t) \) and lateral \( (CL_h) \) clearance ratios of 2.0 and 1.55 and with Reynolds numbers \( (Re) \) ranging from 3400 to 24000. Numerical studies were first performed to validate the experimental results of the circular finned heat sink and reasonably good agreement between the experimental data and numerical results were observed. Comparison of the experimental results showed that the heat transfer performances of the airfoil and rounded rectangular heat sinks exceeded those of the circular heat sink. The experimental Nusselt numbers were computed based on the heat sink base area \( (Nu_b) \) and the total heat transfer area \( (Nu_t) \). In comparison with the circular heat sink, highest enhancements in \( Nu_b \) and \( Nu_t \) of the NACA 0024 heat sink at \( \alpha = 0^\circ \) were 29% and 34.8%, respectively. In addition, the overall heat transfer performances of the NACA 0024 heat sinks were also seen to increase with increasing \( \alpha \). The results suggest that the streamline geometry of the airfoil heat sink has low air flow resistance, which resulted in insignificant bypass effect and thereby improving the heat sink thermal performance. In addition, the increase in \( \alpha \) further improves the heat transfer performance of the NACA 0024 heat sinks through the formation of vortices which enhanced fluid mixing. Finally, based on the above mechanisms proposed, a semi-analytical model was developed to characterize the heat transfer performances of the NACA 0024 heat sinks for the range of \( \alpha \) and \( Re \) tested. In comparison with the experimental data, reasonably accurate predictions were achieved with the model where the deviations in \( Nu_b \) were less than 7% for \( Re \geq 6800 \).

KEY WORDS: Forced convective heat transfer; heat sink; airfoil; selective laser melting

1. Introduction

Forced convective heat transfer with extended surfaces is commonly used to cool electronic devices. Air is often a preferred cooling medium as it is readily available and effective cooling can be achieved without the need of complex operating facilities. Finned arrays are commonly installed on electronic heat sources to maintain the component temperatures within the operating limits. The increase in heat transfer surface area and the induced surface-to-air interactions such as turbulent mixing, vortex shedding and thermal boundary layer disruption are the mechanisms widely suggested for the enhanced thermal performance observed. However, with the miniaturization of electronic devices and the increase in component level heat flux, the continuous development of pin fin designs with greater cooling efficiency becomes increasingly important.

Investigations on pin fin heat sinks can be broadly classified into (1) the effects of pin fin
arrangement, (2) the effects of channel wall-to-fin clearance, and (3) the effects of pin fin geometry. Bilen et al. [1], for instance, investigated the heat transfer performances of in-line and staggered cylindrical fin array heat sinks with varying fin separation in the streamwise direction. It was determined that the staggered arrangement resulted in higher heat transfer enhancement than the in-line arrangement and the maximum heat transfer was recorded with the fin separation-to-diameter ratio of 2.94. Subsequently, Jeng and Tzeng [2] studied the performances of square fin arrays with varying streamwise and spanwise fin separations using the transient single-blow technique. The fin Nusselt number of staggered square fins was determined to be approximately 20% higher than the in-line circular fins and the best performing staggered square fins have inter-fin pitches of 1.5 in both streamwise and spanwise directions. Similar studies were also performed by Akyol and Bilen [3] and

<table>
<thead>
<tr>
<th>Nomenclature</th>
<th>Greek Symbols</th>
<th>Subscripts</th>
<th>Constants</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$ area ($m^2$)</td>
<td>$\alpha$ angle of attack</td>
<td>$ave$ average</td>
<td>$A$ constant in Eqs. (40) and (42)</td>
</tr>
<tr>
<td>$A_b$ base area ($m^2$)</td>
<td>$\epsilon$ turbulent energy dissipation rate ($m^2/s^2$)</td>
<td>$b$ base</td>
<td>$B$ constant in Eqs. (40) and (42)</td>
</tr>
<tr>
<td>$A_t$ total heat transfer area ($m^2$)</td>
<td>$\Gamma$ circulation ($m^2/s$)</td>
<td>$fc$ forced convection</td>
<td>$C_{f}$ constant in Eqs. (23) and (40)</td>
</tr>
<tr>
<td>$AR$ aspect ratio</td>
<td>$v$ kinematic viscosity ($m^2/s$)</td>
<td>$in$ inlet</td>
<td>$C_i$ constant in $k$-$\epsilon$ model</td>
</tr>
<tr>
<td>$B$ heat sink width ($m$)</td>
<td>$v_t$ turbulent kinematic viscosity ($m^2/s$)</td>
<td>$nc$ natural convection</td>
<td>$C_{p}$ constant in $k$-$\epsilon$ model</td>
</tr>
<tr>
<td>$c$ chord length ($m$)</td>
<td></td>
<td>$out$ outlet</td>
<td>$k$ constant in $k$-$\epsilon$ model</td>
</tr>
<tr>
<td>$C$ flow channel cross section height ($m$)</td>
<td></td>
<td>$rad$ radiation</td>
<td>$m$ constant in Eq. (19)</td>
</tr>
<tr>
<td>$C_s$ skin friction drag coefficient</td>
<td></td>
<td>$s$ solid</td>
<td>$n$ constant in Eq. (19)</td>
</tr>
<tr>
<td>$C_p$ pressure drag coefficient</td>
<td></td>
<td>$sim$ simulated</td>
<td>$\sigma_k$ constant in $k$-$\epsilon$ model</td>
</tr>
<tr>
<td>$C_i$ induced drag coefficient</td>
<td></td>
<td>$sys$ system</td>
<td>$\sigma_{e}$ constant in $k$-$\epsilon$ model</td>
</tr>
</tbody>
</table>
| $CL_t$ tip clearance ratio            |                                    | $t$ total         |}

...
Bilen et al. [4] with hollow rectangular and tube fins and the relationships between fin arrangement and heat transfer performance were correspondingly established. In the above studies, staggered arrangements have demonstrated higher heat transfer performances as compared to in-line arrays as a result of the increased turbulence mixing but due to the additional obstruction to the fluid flow imposed by the staggered arrangement, higher pumping power across the heat sinks is also required.

In many practical applications, heat sinks are often mounted on electronic heat sources where they are not fully shrouded. As the fluid tends to seek the path of least resistance, the existence of tip and lateral clearances may result in significant amount of air flow bypassing the fins, reducing the air velocity through the fins and affecting the heat sink performance. Over the years, the effects of clearance ratio on the hydraulic and thermal performances of the heat sinks have also been critically examined. For instance, Sparrow et al. [5] analyzed the laminar heat transfer characteristics of longitudinal fin arrays and determined that in the presence of larger tip clearance and smaller fin spacing, heat transfer by forced convection was negligible along the fins but increased evidently near the fin tip. Dogruoz et al. [6] experimentally investigated the hydraulic resistance and heat transfer characteristics of in-line square fin array with tip clearance-to-fin height ratios ($CL_t$) ranging from 0 to 3 and concluded that the effects of fin geometry on the hydraulic resistance of the heat sinks diminished with increasing $CL_t$. In addition, experiments conducted at the approach velocity of 4 m/s also revealed that the heat sink thermal resistance has relatively low sensitivity to the change in $CL_t$.

In the studies performed by Elshafei [7], the $CL_t$ of 0.22 exhibited higher Nusselt number as compared to the fully-shrouded configuration and it was suggested that the clearance gap served as turbulence promoter to increase the heat transfer rate. In addition to experimental investigations, numerous models which included the effects of bypass were also proposed to predict the heat sinks' performance. For instance, Jonsson and Moshfegh [8] investigated the performance of heat sinks with different tip and lateral clearances and developed empirical bypass correlations to predict the heat sinks’ Nusselt number and dimensionless pressure drop. On the other hand, Dogruoz et al. [6] developed a semi-analytical two-branch by-pass model by assuming that the air flow through the heat sink does not change along the streamwise direction and subsequently included the effects of air leakage from the heat sink [9]. In the above models, friction factor correlations were introduced to obtain the pressure differences across the flow channel and thereby, the average velocity and heat transfer from the heat sink were determined. A similar approach was also employed by Khan et al. [10] to evaluate the effects of tip and lateral clearances on the thermal and hydraulic performances of cylindrical fin array under laminar forced convection and it was shown that clearance ratios significantly reduced pressure drop and heat dissipated from the heat sink.

While it is intended for the heat transfer coefficients of heat sinks to be maximized, for efficient heat dissipation, the enhanced thermal performances should be achieved with minimal increase in system pressure. Apart from configuring the fin arrangements and flow channel sizes as discussed above, fins of different geometries have also been explored to optimize the heat sinks’ heat transfer and hydraulic performances and their efficiencies have been evaluated. For instance, Sparrow and Grannis [11] performed experimental and numerical studies to characterize the pressure drop across diamond-shaped pin fin arrays. Subsequently, Tanda [12] determined experimentally that the diamond-shaped fins enhanced heat transfer by up to 1.65 times as compared to an unfinned channel at constant pumping power. Sparrow et al. [13] consolidated and compared heat transfer performances of fins of different cross-sectional geometries. Based on their analysis, it was suggested that under high Re, heat transfer from non-circular cylinders, such as fins of square, diamond and hexagonal shaped cross-sections, was enhanced by circulation of air in the regions of the fins that were experiencing flow separation. More recently, Tong et al. [14] extended the studies to include chamfered cylinders of various angles of attack. Correlations of Nu for chamfered cylinders at various Re and angles of attack were developed and the results obtained for air were also extended to other fluids. Lately, Fan et al. [15] developed a novel cylindrical oblique fin heat sink which decreased the total thermal resistance by up to 59.1% with negligible pressure difference as compared to conventional straight fins and suggested that the enhanced thermal performance was due to the disruption and initialization of boundary layer at the leading edge of each fin. The use of streamline shaped fins such as elliptical, drop-shaped and airfoil fins to reduce the flow resistance and increase heat transfer across the flow
channel is another viable solution which have been investigated experimentally and numerically. For instance, experimental results obtained by Chen et al. [16] showed that the drop-shaped fins have higher heat transfer than circular fins with much lower resistance and similar conclusions for elliptical fins were obtained by Li et al. [17]. On the other hand, Wang et al. [18, 19], provided some insights on the forced convective heat transfer performances of an internally heated NACA 63421 airfoil. It was shown from their experiments that the Nusselt number increases at higher angles of attack and the modified Hilpert correlation [18] was proposed to characterize the measured airfoil data. Sahiti et al. [20] numerically investigated the pressure and heat transfer performances of six different forms of pin fin cross-sections. Elliptic and NACA profile of 0.5 thickness-to-chord length ratio, along with other streamline and conventional geometries were examined. The simulation results showed that the elliptic profile performed better than other cross-sections whereas the NACA profile did not show significant advantage due to the low Reynolds number simulated and the large thickness-to-chord length ratio of the airfoil. On the other hand, pin fins of NACA 0050 profiles (thickness-to-chord length ratio of 0.5) were studied by Zhou and Catton [21]. The plate-pin fin heat sinks (PPFHSs) were simulated at velocities ranging from 6.5 m/s to 12.2 m/s. Their results showed that heat transfer effectiveness factors of the NACA 0050 fins were comparable to that of elliptic fins.

Apart from the commonly used manufacturing techniques such as die-casting, extrusion and injection molding, recent developments in Selective Laser Melting (SLM) technology also offer an alternative approach in heat sink fabrication. Selective Laser Melting is a branch of additive manufacturing technique which utilizes a high-power laser source to melt and fuse the base metal powder in accordance to pre-programmed models where complex three-dimensional structures can be achieved by melting consecutive layers of base powder over each other. Ventola et al. [22] used the direct metal laser sintering technique and created heat sinks of different artificial surface roughness. From their experiments, a peak enhancement of 73% in the Nusselt number of their sample surface as compared to that of the smooth surfaces was determined. Wong et al. [23], on the other hand, fabricated two conventional and three novel heat sinks using the SLM technique. Complex designs such as the lattice structure and the elliptical array were successfully produced. While it was shown that the lattice structure demonstrated poor heat transfer and low flow resistance, the elliptical array exhibited the most efficient performance with the highest heat transfer rate per unit pressure drop.

As shown in the above brief review, heat sinks with streamline shaped pin fin arrays have achieved relative success in enhancing heat transfer efficiencies. Under conditions where heat sinks are not fully shrouded, streamline geometries such as airfoil shaped pin fins also offered low flow resistance which reduces the amount of air bypass and enhanced cooling by allowing higher air flow through the fin array. However, experimental results with airfoil heat sinks are scarce and numerical studies performed are also limited to airfoils with large thickness-to-chord length ratio. In the present study, SLM was employed to fabricate staggered arrays of streamline airfoil heat sinks with a small thickness-to-chord length ratio of 0.24 and with different angles of attack. In addition, heat sinks of staggered circular and rounded rectangular fin array were also fabricated for comparison. The thermal characteristics of the heat sinks were investigated in a rectangular flow channel with tip and lateral clearance ratios of 2.0 and 1.55, respectively to simulate practical applications where the heat sinks are not fully shrouded. Finally, based on the experimental results obtained, the thermal transport mechanisms associated with the airfoil heat sinks are elucidated and a semi-analytical heat transfer model which incorporated the effects of these mechanisms is also proposed.

2. Surfaces preparation and characterization

The SLM 250 HL (SLM Solutions GmbH) facility at the Future of Manufacturing Laboratory 1 of Singapore Centre for 3D Printing (SC3DP) in Nanyang Technological University (NTU), Singapore was employed to fabricate the heat sinks in the present investigation. The machine which consists of a Gaussian distributed Yb:YAG laser with a maximum power of 400 W and a laser beam spot size of 80 µm was utilized to melt and fuse the base AlSi10Mg powder of 20 µm to 63 µm size distribution. AlSi10Mg was selected due to its light weight (with density of approximately 2670 kg/m³ [24]), and
high thermal conductivity (up to 175 W/m·K [24]), which make it suitable for electronic cooling applications. The laser melting process was carried out in the machine’s build chamber where inert argon gas was first used to flush the chamber to attain an oxygen level of less than 0.2% so as to minimize oxidation and combustion of powder. Subsequently, the first layer of AlSi10Mg metallic powder was distributed evenly on the base-plate by a recoater and the laser beam was directed to melt the powder based on a preprogrammed model. Upon completion of the laser melting process for the first layer, the base-plate was then lowered by one-layer thickness of 50 µm and the process was repeated until the parts are fully constructed. In the present investigation, the laser power, scanning speed and hatching spacing of 350 W, 1150 mm/s and 0.17 mm were respectively selected. In the present study, heat sinks which are staggered arrays of NACA 0024 pin fins at angles of attack (α) = 0°, 5°, 10°, 15° and 20° and NACA 4424 pin fins at α = 0° were produced. In addition, pin fin heat sinks with circular and rounded rectangular geometries were also fabricated and served as comparison against the airfoil heat sink.

Each heat sink consists of a square base plate of 50 mm × 50 mm with thickness of 5 mm. The pin fins are integrated onto the base plate in a single built piece with a constant fin height of 25 mm. Schematic diagrams of the circular, rounded rectangular, NACA 0024 (α = 0°) and NACA 4424 (α = 0°) heat sinks are shown in Fig. 1. For the circular heat sink, each pin fin has a diameter of 4 mm with separations between adjacent fins in the spanwise (S_x) and streamwise (S_y) directions of 5 mm each. The rounded rectangular heat sink, on the other hand, consists of circular edges of 2 mm diameter and length and width of 6 mm and 2 mm, respectively. Similar to the circular heat sink, S_x and S_y between adjacent rounded rectangular fins are also fixed at 5 mm. For all the airfoil heat sinks, the maximum thickness and chord length of the airfoil profile were fixed at 2.4 mm and 10 mm which resulted in the ratio of 0.24. Each NACA 0024 pin fin has a profile which is symmetrical about the centerline of the airfoil whereas the NACA 4424 has an asymmetrical profile with 4% camber located at 40% chord length from the airfoil leading edge. On the other hand, the NACA 0024 heat sinks with α = 5°, 10°, 15° and 20° have the pin fins oriented at angles relative to the air flow direction and are symmetrical about the centerline of the heat sink. For all the airfoil heat sinks, S_x is similarly fixed at 5 mm whereas S_y is approximately 10 mm, where S_x and S_y are measured from the 50% chord length to the same chord length position of its adjacent airfoil. Schematics of the NACA 0024 α = 10° and 20° are depicted in Fig. 2 and dimensions of the all sink heats investigated are summarized in Table 1.

![Fig. 1 Top views of (a) circular, (b) rounded rectangular, (c) NACA 4424 (α = 0°) and (d) NACA 0024 (α = 0°) heat sinks.](image1)

![Fig. 2 Top views of NACA 0024 heat sinks at (a) α = 10° and (b) α = 20°.](image2)
Table 1 Summary of heat sink dimensions

<table>
<thead>
<tr>
<th>Heat sink</th>
<th>Pin fin description</th>
<th>No. of fins</th>
<th>Spanwise separation, ( S_x ) (mm)</th>
<th>Streamwise separation, ( S_y ) (mm)</th>
<th>Total heat transfer area, ( A_t ) (m²)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular</td>
<td>• Diameter, 4 mm</td>
<td>41</td>
<td>5</td>
<td>5</td>
<td>0.01538</td>
</tr>
<tr>
<td>Rounded Rectangular</td>
<td>• Length, 6 mm • Width, 2 mm • Rounded edge diameter, 2 mm</td>
<td>41</td>
<td>5</td>
<td>5</td>
<td>0.01714</td>
</tr>
<tr>
<td>NACA 4424 (( \alpha = 0^\circ ))</td>
<td>• Chord length, 10 mm • Thickness, 2.4 mm</td>
<td>23</td>
<td>5</td>
<td>10</td>
<td>0.01475</td>
</tr>
<tr>
<td>NACA 0024 (( \alpha = 0^\circ ))</td>
<td>• Chord length, 10 mm • Thickness, 2.4 mm</td>
<td>23</td>
<td>5</td>
<td>10</td>
<td>0.01471</td>
</tr>
<tr>
<td>NACA 0024 (( \alpha = 5^\circ ))</td>
<td>• Chord length, 10 mm • Thickness, 2.4 mm</td>
<td>23</td>
<td>5</td>
<td>10</td>
<td>0.01471</td>
</tr>
<tr>
<td>NACA 0024 (( \alpha = 10^\circ ))</td>
<td>• Chord length, 10 mm • Thickness, 2.4 mm</td>
<td>23</td>
<td>5</td>
<td>10</td>
<td>0.01471</td>
</tr>
<tr>
<td>NACA 0024 (( \alpha = 15^\circ ))</td>
<td>• Chord length, 10 mm • Thickness, 2.4 mm</td>
<td>23</td>
<td>5</td>
<td>10</td>
<td>0.01471</td>
</tr>
<tr>
<td>NACA 0024 (( \alpha = 20^\circ ))</td>
<td>• Chord length, 10 mm • Thickness, 2.4 mm</td>
<td>23</td>
<td>5</td>
<td>10</td>
<td>0.01471</td>
</tr>
</tbody>
</table>

Photographs of the fabricated circular, rounded rectangular, NACA 0024 and NACA 4424 heat sinks are depicted in Fig. 3. Using the OLYMPUS SZX7 Stereo Microscope, up to 20 measurements for each dimension of all the fabricated heat sinks were taken and it was determined that the deviations between the as-built and design dimensions range between 0.6% and 2.5%.

![Fig. 3 Photographs of (a) circular, (b) rounded rectangular, (c) NACA 4424 and (d) NACA 0024 heat sinks.](image-url)

3. Experimental setup and procedures

3.1 Experimental setup

A schematic diagram of the experimental setup used is shown in Fig. 4. The flow channel is vertically mounted and has a total length of 1000 mm, from the air flow inlet to the fan discharge. Based on the flow channel cross-sectional dimensions of 127.5 mm × 75 mm, the hydraulic diameter \( (D_h) \) is computed to be 94.4 mm. A suction fan is located at the outlet of the flow channel and is connected to a variable speed drive which enables the air velocity through the flow channel to be controlled during the experiments. An air straightener is installed immediately after the flow channel inlet and the center of the test section, where the heat sink is mounted, is located 550 mm from the inlet. Two K-type thermocouples are used to measure the mean inlet \( (T_{in}) \) and outlet \( (T_{out}) \) air
temperatures and are located at 100 mm from the inlet and 180 mm from the exit, respectively. The thermocouple port which measures the outlet air temperature is designed such that the thermocouple is allowed to traverse in the axis perpendicular to the air flow direction to obtain the outlet air temperatures at various locations. Four outlet air temperatures were measured at 10 mm, 30 mm, 50 mm and 70 mm from the channel base and the average were taken as $T_{out}$. An air flow sensor is positioned at 350 mm from the inlet to obtain the average air velocity ($U$), where the Reynolds number ($Re$) is then determined using Eq. (1). A manometer is used to measure the overall system pressure ($P_{sys}$) with the pressure port located 650 mm downstream of the inlet. Figure 6 shows the cross-sectional dimensions of the flow channel where $C$ and $W$ indicate the height and width of the flow channel and $H$ and $B$ represent the height and width of the heat sink. Using Eqs. (2) and (3), the tip ($CL_t$) and lateral ($CL_h$) clearance ratios are calculated as 2.0 and 1.55, respectively.

$$Re = \frac{\rho U D_h}{\mu}$$  \hspace{1cm} (1)

$$CL_t = \frac{C}{H} - 1$$  \hspace{1cm} (2)

$$CL_h = \frac{W}{B} - 1$$  \hspace{1cm} (3)

Fig. 4 Schematic of experimental setup.
Details of the test section are also shown in Fig. 5. Four cartridge heaters of diameter 3.18 mm and length 32 mm are inserted into copper block of 50 mm × 50 mm × 20 mm. The heat sink to be tested is adhered to the copper block using a thermal conductive epoxy. The cartridge heaters are connected to a variable AC power transformer which is employed to provide the required power input to the heaters. Heat is conducted from the heaters through the copper block to the heat sink. In order to prevent excessive heat losses, a Teflon block (80 mm × 80 mm × 57 mm) is used as the first layer of insulation which encloses the cartridge heaters, the copper block and the parameter of the heat sink base plate. The assembly is then fitted into an aluminum holder which is in turn secured onto the bottom side of the air flow channel such that the top surface of the heat sink base plate flashes with the internal channel wall. Finally, a 6-mm-thick foam made of elastomer, which serves as a second layer of insulation, is applied around the aluminum holder.

In total, 12 K-type thermocouples are used to determine the heat sink’s surface temperature and to estimate the overall heat losses from the heater section. Two thermocouples (T1 and T2) as shown in Fig. 5 are inserted into the heat sink base plate and at approximately 2 mm from the top surface of the base plate. The base surface temperature of the heat sink (Ts) is obtained by averaging the temperatures from T1 and T2. In addition, four thermocouples (T3, T4, T5 and T6) are inserted into the copper block at equal distances from each other and at 5 mm from the top of the copper block, to monitor the copper block temperature throughout the experiments. Another four thermocouples (T7, T8, T9 and T10), fitted on the backside of the heater section, are used to determine the surface temperature of the insulation, for the computation of heat losses through natural convection (ql,nc). Finally, thermocouples T11 and T12 are inserted at 2 mm from the top of the Teflon block to obtain the average Teflon surface temperature where the heat losses through forced convection (ql,fc) are estimated. Prior to the experiments, the thermocouples were calibrated using the 7103 Micro-Bath Thermometer Calibrator.

3.2 Experimental procedures and data reduction

For each heat sink, experiments were conducted at the constant input heat rate (qt) of 10 W and air velocities between 0.5 and 3.5 m/s (3400 ≤ Re ≤ 24000). The variable transformer was employed to provide the required power input to the cartridge heaters and the fan’s variable speed drive was utilized to control the air flow rate through the flow channel. Experiments were performed by varying the air flow velocity at intervals of 0.5 m/s and at each interval, the waiting time was between 30
minutes and 2 hours for steady state to be achieved, where the temperature reading fluctuations were within ± 0.1°C for approximately 5 minutes. Thereafter, the respective temperatures of the thermocouples were recorded using the MW100 Data Acquisition Unit at the sampling rate of 2 Hz over 1 minute. In addition, \( P_{sv} \) were also recorded from the inclined manometer. After completing the above cycle, the experiments were then repeated.

The forced convection heat transfer of the heat sink \( (q_{fc}) \) can be obtained from Eq. (4), where \( q_{l,rad} \) is the radiation heat loss and the other terms are as described above. Using the heat sink base temperature recorded \( (T_1–T_2) \) and by assuming the emissivity value of 0.09, which is similar to commercial aluminum, \( q_{l,rad} \) of the heat sink was computed to be less than 1% of \( q_t \). In addition, using the temperature of the elastomer foam measured by thermocouples \( T_7–T_{10} \), the maximum radiation heat loss from the backside of the heater section was calculated to be less than 0.03% of \( q_t \). As these radiation heat losses are small, they are neglected in the computation of \( q_{fc} \). On the other hand, \( q_{l,nc} \) from the back side of the heater section was computed with \( T_7 \) to \( T_{10} \) obtained from the experiments and by applying natural convection correlations for vertical, bottom and top walls given by Incropera et al. [25]. Finally, based on the numerical simulation results for circular pin fin heat sink (see Section 4.1 for details) the heat losses through forced convection from the top Teflon surface was correlated as a function of \( Nu, Re \) and \( Pr \) as shown in Eq. (5). Using the Teflon surface temperatures recorded by thermocouples \( T_{11} \) and \( T_{12} \) and Eq. (5), \( q_{l,fc} \) was determined.

\[
q_{fc} = q_t - q_{l,nc} - q_{l,fc} - q_{l,rad}
\]

\[
Nu_{tg} = 0.135Re^{0.6263}Pr^{0.33}
\]

Based on the heat losses determined for each sink heat and at the respective Re, the computed \( q_{fc} \) values were then used to obtain the forced convective heat transfer coefficient with respect to the base plate area \( (h_b) \) and the total heat transfer area \( (h_t) \). In addition, as the air is heated as it traverses across the heat sink, the average air temperature, \( \frac{T_{in} + T_{out}}{2} \), is used to evaluate \( h_t \) and \( h_b \) as shown in Eqs. (6) and (7), respectively.

\[
h_b = \frac{q_{fc}}{A_b \left(T_s - \frac{T_{in} + T_{out}}{2}\right)}
\]

\[
h_t = \frac{q_{fc}}{A_t \left(T_s - \frac{T_{in} + T_{out}}{2}\right)}
\]

Using Eqs. (8) and (9), \( Nu_b \) and \( Nu_t \) are computed as follows:

\[
Nu_b = \frac{h_b D_h}{k_l}
\]

\[
Nu_t = \frac{h_t D_h}{k_l}
\]

The uncertainties of the current and voltage readings from the variable transformer are within ± 0.5% of their full scale whereas the thermocouples were calibrated to within ± 0.5°C deviation for the range of temperatures tested. The accuracy of the air flow sensor is within ± 3% of its full scale and the inclined manometer has the accuracy of ± 0.5 Pa. Throughout the experiments, the fluid-to-wall temperature difference ranged between 12.2°C and 35.7°C whereas the air flow velocities are between 0.5 and 3.5 m/s. Using the method described by Moffat [26], the average uncertainties of \( h, Nu_b \) and \( Re \) were determined to be 6.6%, 6.6% and 7.8%, respectively and the maximum uncertainties of \( h, Nu_b \) and \( Re \) are 7.5%, 7.5% and 21%, respectively.
4. Results and discussions

4.1 Validation of experimental data

In order to validate the accuracy of the experimental data collected, numerical simulations were performed on the circular pin fin heat sink. As the experiments were conducted for Re > 2300, turbulent flow was assumed and the three-dimensional, steady state, time averaged continuity and momentum equations for incompressible flow are employed as shown in Eqs. (10) and (11). In addition, the standard k-ε turbulence model [Eqs. (12) - (14)] was also adopted to approximate the Reynolds stresses. Due to the thermal interactions between the heat sink and mainstream air, a conjugate heat transfer model which governs the fluid [Eq. (15)] and solid [Eq. (16)] domains was set up with coupled boundary conditions at the solid/liquid interface. In the equations below, $C_\mu$, $C_1$, $C_2$, $\sigma_k$ and $\sigma_e$ are empirical constants with the values of 0.09, 1.44, 1.92, 1.0 and 1.3, respectively.

\[
\frac{\partial \bar{U}_i}{\partial x_i} = 0 \tag{10}
\]

\[
\bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_j^2} + \frac{\partial \bar{U}_i \bar{u}_j'}{\partial x_j} \tag{11}
\]

\[
\bar{U}_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \nu_t \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{u}_j'}{\partial x_i} \right) \frac{\partial \bar{U}_i}{\partial x_j} - \varepsilon \tag{12}
\]

\[
\bar{U}_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \left( \nu + \frac{\nu_t}{\sigma_e} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + C_1 \frac{\varepsilon}{k} \nu_t \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{u}_j'}{\partial x_i} \right) \frac{\partial \bar{U}_i}{\partial x_j} - C_2 \frac{\varepsilon^2}{k} \tag{13}
\]

\[
\nu_t = \frac{C_\mu k^2}{\varepsilon} \tag{14}
\]

\[
\bar{U}_j \frac{\partial \bar{T}}{\partial x_j} = \left( \frac{\nu}{Pr} + \frac{\nu_t}{Pr_t} \right) \frac{\partial^2 \bar{T}}{\partial x_j^2} \tag{15}
\]

\[
k_{ls} \frac{\partial^2 \bar{T}_s}{\partial x_j^2} + \dot{q} = 0 \tag{16}
\]

The computational domain is shown in Fig. 6 (a) and details of the simulated heater section are depicted in Fig. 6 (b). For the simulation, radiation heat loss was neglected and ambient pressure was assumed at the flow channel inlet with the constant inlet air temperature ($T_{in}$) of 30°C whereas at the outlet, uniform air velocity ($U_{out}$) was employed. In addition, no slip boundary condition was applied and the outer walls of the flow channel were assumed to be thermally insulated. The simulated heat sink and the heater section are similar to those used in the experiments. Finally, to save computational time, the simulated power output from the cartridge heaters ($q_{sim}$) was taken to be $q_{sim} = q_t - q_{l,nc}$, where $q_t = 10$ W and $q_{l,nc}$ was obtained from the experiments. Hence, this allows the natural convection heat losses from the back of the heater section to be neglected from the simulations. The boundary conditions as mentioned above are shown in Eqs. (17) – (21). The simulations were performed using the “Comsol Multiphysics” software and the unstructured grid system of different mesh elements was used. The convergence criterion was set at 0.001 and mesh independence tests were conducted where the results were obtained with approximately 1,000,000 mesh elements.
\[ T(0,y,z) = T_{in} \]  
\[ U_x(L,y,z) = U_{out} \quad U_y(L,y,z) = 0 \quad U_z(L,y,z) = 0 \]  
\[ U_l(x,0,z) = U_l(x,W,z) = U_l(x,y,0) = U_l(x,y,C) = 0 \]  
\[ \frac{\partial T}{\partial y} (x,0,z) = \frac{\partial T}{\partial y} (x,W,z) = \frac{\partial T}{\partial z} (x,y,0) = \frac{\partial T}{\partial z} (x,y,C) = 0 \]  
\[ \dot{q} = q_{sim} = q_t - q_{Lnc} \]  

Figure 6 (a) Simulation domain and (b) circular heat sink and heater section.

Figure 7 (a) shows the simulation results of the circular heat sink at \( Re = 24000 \) (\( U = 3.5 \) m/s). It can be observed that surface temperature of the fins increases with decreasing fin height and the highest temperature is obtained at the fin base. In addition, the base and fin temperatures of the heat sink are also seen to increase in the streamwise direction. On the other hand, side view of the air flow velocity profile across the heat sink is depicted in Fig. 7 (b). As the air traverses downstream of the heat sink, it is seen that the air velocity along the top bypass section increases while the air velocity through the heat sink reduces significantly. The top view of the air flow velocity profile and streamline patterns are shown in Figs. 7 (c) and (d), respectively. From Figs. 7 (c) and (d), it can be observed that, apart from top bypass, there was also significant side bypass. In seeking the path of least resistance, the air tends to also escape from the sides of the heat sink and there is substantial reduction of air flow velocity from the 5th row of fins onward. In addition, from the streamline patterns shown in Fig. 7 (d), it can also be observed that flow separation occurred at the rear of each circular fin which resulted in slightly higher temperature at the rear of the fin as compared to the frontal fin region where the mainstream air directly impinges. Finally, it can also be seen that small scale vortices were generated on the few fins that were located at the upstream edge of the heat sink. Conventionally, vortices were suggested to increase turbulence mixing and improve heat transfer of the heat sink. However, as the magnitude and quantities of the vortices that were generated on the circular heat sink were small, their effects on enhancing the heat transfer performance of the heat sink are not significant. In all, the simulation results suggest that the circular fin structure, with poor hydraulic performance, have resulted in high flow resistance and undesirable flow separation. Hence, due to the reduction in air flow rate downstream of the heat sink, convective cooling correspondingly decreased, resulting in the increasing fin and base temperatures in the streamwise direction [Fig. 7 (a)].
Fig. 7 (a) Surface temperature distribution of circular heat sink in °C, (b) side view of air flow velocity profile over circular heat sink in m/s, (c) top view of air flow velocity profile over circular heat sink in m/s and (d) top view of streamline patterns (symmetry from mid-point up) in m/s at Re = 24000.

With the simulation results, the forced convection heat losses from the top Teflon surface \( (q_{Tfc}) \) at different Re were obtained and the respective Nu_{Tfc} were correlated as shown in Eq. (5). In addition, using the simulated average based temperature of the circular heat sink and Eq. (8), Nu_{b} was calculated and comparisons against the present experimental results are plotted in Fig. 8 (a). As shown in Fig. 8 (a), the simulated Nu_{b} values agree well with the experimental results, with a deviation of lower than 6.2% for Re ≥ 10000. However, at low Re (< 10000) larger discrepancies between the
experimental data and simulation results can be observed. Due to the laser melting process, it can be observed that the heat sinks have high surface roughness [Fig. (3)]. In a recent investigation by Ho et al. [27], it was reported that the SLM fabricated surface has a root mean square (rms) roughness of 7.32 μm. In comparison, commercially available plain Al-6061 has an rms value of only 0.25 μm. It is likely that, even at low Re values, significant turbulence have been induced due to high surface roughness which improved the heat sink heat transfer performance and hence, explains the under-prediction of the k-ε model for Re < 10000. In all, for the circular heat sink, the experimental results agree reasonably well with the simulated Nu_b values with the average discrepancy in Nu_b of approximately 10%. In addition, the system pressures (P_sys) of the flow channel with the circular heat sink were also measured and comparison against the results from the simulation is shown in Fig. 9 (b). Since the low system pressure, P_sys for Re < 10,000 would result in significantly large uncertainties they are not presented. For Re ≥ 10000, it can be seen that the experimental P_sys values agree well with the simulation result and the maximum discrepancy in P_sys is approximately 4.6%.

Fig. 8 (a) Comparison of circular heat sink Nu_b obtained from experiments and simulations.

Fig. 8 (b) Comparison of P_sys of circular heat sink obtained from experiments and simulations.
4.2 Heat transfer performances of SLM fabricated heat sinks

4.2.1 Effects of fin geometry

The experimental $\text{Nu}_b$ of the circular, rounded rectangular, NACA 4424 and NACA 0024 heat sinks at $\alpha = 0^\circ$ are presented in Fig. 9. The rounded rectangular, NACA 0024 and NACA 4424 heat sinks demonstrated consistently higher thermal performances as compared to circular heat sink for the range of Re tested. As compared to the circular heat sink, the highest enhancements in $\text{Nu}_b$ of 34.7%, 29% and 28.5% were respectively recorded for rounded rectangular, NACA 0024 and NACA 4424. In addition, it can be observed that the enhancements in $\text{Nu}_b$ for the NACA 0024 and NACA 4424 heat sinks decrease with increasing Re whereas for the rounded rectangular heat sink, enhancements in $\text{Nu}_b$ remain relatively constant as Re changes. In order to evaluate the overall performances of the heat sinks, the average $\text{Nu}_b$ ($\text{Nu}_{b,\text{ave}}$) of the respective heat sinks were calculated by averaging $\text{Nu}_b$ for the range of Re tested. On the basis of this approach, the corresponding enhancements in $\text{Nu}_{b,\text{ave}}$ as compared to circular heat sink were calculated to be 32.5% for rounded rectangular, 11.5% for NACA 0024 and 10.9% for NACA 4424.

As shown in Table 1, due to the differences in fin geometries, the total heat transfer areas ($A_t$) of the heat sinks are also different. An accurate analysis of the heat sinks’ thermal performance which accounts for the differences in $A_t$ can then be obtained by comparing the Nusselt numbers of the heat sinks that are normalized by $A_t$ (or $\text{Nu}_b$) as shown in Eqs. (7) and (9). $\text{Nu}_t$ of the heat sinks are plotted against Re in Fig. 10. In comparison with circular heat sink, the highest enhancements in $\text{Nu}_t$ and the enhancements in $\text{Nu}_{t,\text{ave}}$ are 20.8% and 18.9% for the rounded rectangular heat sink, 34.8% and 16.6% for the NACA 0024 heat sink, and 34% and 15.6% for the NACA 4424 heat sink, respectively. At low Re, $\text{Nu}_t$ of both airfoil heat sinks are marginally higher than the rounded rectangular heat sink. For instance at Re = 3400, $\text{Nu}_t$ of NACA 0024 and NACA 4424 are 13.3% and 12.6% higher than the rounded rectangular heat sink. However, as Re increases, $\text{Nu}_t$ of the rounded rectangular heat sink are observed to outperform the airfoil heat sink and as high as 11.4% higher $\text{Nu}_t$ as compared to NACA 4424 were recorded at Re = 24000.

In comparison with circular heat sink, the airfoil heat sinks with streamline geometries had much lower air flow resistance. Hence, for the same Re, the airfoil geometries resulted in minimal air bypass and enabled higher air flow rate through the heat sink, which in turn increased the heat...
removal rate and explains the highest \( \text{Nu} \) amongst all the heat sinks at low Re. However, as air flow velocity increases with increasing Re, due to the blunt edges of the rounded rectangular heat sink, vortices were likely to be formed at the trailing edge of the fins which induced fluid mixing and further enhanced heat transfer. This explanation agrees well with the recent airflow visualization studies conducted by Wong et al. [28] where vortices were observed to be generated at the trailing edge of a rounded rectangular fin. On the other hand, poorer heat transfer of the NACA 0024 and NACA 4424 heat sinks as compared to the rounded rectangular heat sink also suggest that the streamline characteristics of the airfoil were incapable of inducing vortices along the fins at \( \alpha = 0^\circ \).

Apart from the formation of vortices along the fins, the formation of horseshoe vortices at the endwalls is also a commonly observed phenomenon. Several investigations have also shown that the presence of horseshoe vortices enhances the local heat transfer coefficient at the leading edge of the fin base [29-31]. Horseshoe vortices were likely to be generated at the endwalls of all the heat sinks used in the present study. However, due to the blunt geometry of the circular heat sink which depletes air flow downstream of the heat sink, horseshoe vortices were likely to form only on the upstream fins. On the other hand, due to their streamline geometry, airfoil heat sinks reduce air bypass and hence, allow the formation of horseshoe vortices also on the downstream fins. Despite more horseshoe vortices being generated, it should be noted that, at high Re, the vortex generation along the fin tip remains the dominant heat transfer enhancement mechanism as \( \text{Nu} \) of the rounded rectangular heat sink surpasses those of the NACA0024 and NACA4424 heat sinks.

Finally, it should also be noted that insignificant differences in thermal performances between the NACA 0024 and NACA 4424 heat sinks were observed.

![Fig. 10 Nu vs Re of SLM fabricated heat sinks of various geometries.](image_url)

**4.2.2 Effects of angle of attack**

In the previous section, experimental results suggest that while the airfoil heat sinks with \( \alpha = 0^\circ \) enhance the heat transfer by reducing air bypass, their thermal performances were limited by their inability to generate vortices along the fins. However, for a symmetrical airfoil, it has also been established that by increasing \( \alpha \), vortices can be induced and the circulation strength (\( \Gamma \)) can be related to the velocity over the airfoil as shown in Eq. (22). In the effort to validate this hypothesis, the NACA 0024 heat sinks with \( \alpha \) ranging from 0° to 20° were investigated and their thermal performances (\( \text{Nu}_b \) and \( \text{Nu}_t \)) at various Re are presented in Figs. 11 and 12.
\[ \Gamma = \pi aceU \]  

(22)

As shown in Figs. 11 and 12, the increase in \( \alpha \) from 0\(^\circ\) to 5\(^\circ\) resulted in insignificant differences in \( \text{Nu}_b \) and \( \text{Nu}_t \). However, with further increments in \( \alpha \) to 10\(^\circ\) and above, noticeable leftward shifts in the \( \text{Nu}_b \) and \( \text{Nu}_t \) curves were obtained. In comparison with the circular heat sink, NACA 0024 heat sinks with \( \alpha = 10^\circ, 15^\circ \) and \( 20^\circ \) resulted in the highest enhancements of 27.6%, 22.6% and 24.7% in \( \text{Nu}_b \) and the highest enhancements of 33.4%, 29.2% and 29.7% in \( \text{Nu}_t \), respectively. The enhancements in \( \text{Nu}_{t,\text{ave}} \) and \( \text{Nu}_{b,\text{ave}} \) of the NACA 0024 at different \( \alpha \) are plotted in Fig. 13, where \( \text{Nu}_{t,\text{ave}(c)} \) and \( \text{Nu}_{b,\text{ave}(c)} \) denote the \( \text{Nu}_{t,\text{ave}} \) and \( \text{Nu}_{b,\text{ave}} \) of the circular heat sink, respectively. As depicted in Fig. 13, it was evident that the change in \( \alpha \) from 0\(^\circ\) to 5\(^\circ\) had less significant effect on the thermal performance of the airfoil heat sink whereas enhancements in \( \text{Nu}_{t,\text{ave}} \) and \( \text{Nu}_{b,\text{ave}} \) were observed to increase more significantly as \( \alpha \) increased to 10\(^\circ\). However, with further increment in \( \alpha \) to 15\(^\circ\) and 20\(^\circ\), the rate of enhancements in \( \text{Nu}_{t,\text{ave}} \) and \( \text{Nu}_{b,\text{ave}} \) reduces. In all, the enhancements in \( \text{Nu}_{t,\text{ave}} \) of the NACA 0024 heat sinks with \( \alpha = 10^\circ, 15^\circ \) and \( 20^\circ \) as compared to the circular heat sink were computed to be 17.2%, 18.1% and 19.7% whereas the corresponding enhancements in \( \text{Nu}_{t,\text{ave}} \) were 22.5%, 23.46% and 25.2%, respectively. Finally, even though it is not shown in graphs, it should be noted that the \( \text{Nu}_{t,\text{ave}} \) values of the NACA 0024 heat sinks with \( \alpha = 10^\circ, 15^\circ \) and \( 20^\circ \) also surpass that of the rounded rectangular heat sink.
As described by Eq. (22), $\alpha$ is directly proportional to $\Gamma$ which is in turn related to the vorticity of the air flowing over the airfoil. For the NACA 0024 heat sink, the increase in heat transfer performance with increasing $\alpha$ suggests the possibility of heat transfer enhancement due to the formation of vortices over the airfoil. However, as observed from the experimental results, at a low angle of attack ($\alpha = 5^\circ$), it is likely that the range of air flow velocities tested were unable to generate sufficient circulation to significantly influence the thermal performance of the heat sink. On the other hand, when $\alpha$ increases, the magnitude of heat transfer enhancement was restricted by the corresponding increase in the airfoils’ flow resistance, giving rise to higher air bypass and lower air flow through the heat sink. In the present study, the optimal angle of attack was determined to be $\alpha = 10^\circ$ corresponding to the highest rates of increments in $\text{Nu}_{\text{ave}}$ and $\text{Nu}_{\text{b,ave}}$.

Fig. 13 Enhancements in $\text{Nu}_{\text{ave}}$ and $\text{Nu}_{\text{b,ave}}$ of the NACA 0024 heat sinks at different $\alpha$. 

Fig. 12 $\text{Nu}_t$ vs Re of SLM fabricated NACA 0024 heat sinks with different $\alpha$. 

![Diagram showing Nu vs Re for different angles of attack](image-url)
4.3 Semi-analytical model for NACA 0024 heat sinks

In this section, a semi-analytical model which predicts the Nu\textsubscript{b} of the NACA 0024 heat sinks at various Re and α is proposed. For forced convection heat transfer, Nu\textsubscript{b} is a function of Re\textsubscript{1} and Pr and can be correlated as shown in Eq. (23), where \( C_r \) and \( m \) are constants to be determined and, when air is used as the heat transfer medium, \( n \) typically takes value of 0.33. In order to account for the effects of air bypass, Re\textsubscript{1} is determined by using the average air velocity through NACA 0024 heat sinks (\( U_1 \)) and is represented by Eq. (24). In addition, experimental results also suggest that the heat sink thermal performance which is dependent on the vortices generated by the airfoil and can be expressed by Eq. (22). Hence, to account for the effects of vortex generation, it is assumed that \( C_r \) is a function of the dimensionless form of Γ, which itself is a function of α, as shown in Eq. (25).

\[
\text{Nu}_{b} = C_r \text{Re}_1^m \text{Pr}^n \tag{23}
\]

\[
\text{Re}_1 = \frac{\rho U_1 D_{h,1}}{\mu} \tag{24}
\]

\[
C_r = f \left( \frac{\Gamma}{c U_1} \right) = f (n \alpha) \tag{25}
\]

To determine \( U_1 \), the bypass models similar to those presented by Jonsson and Moshfegh [8], Dogruoz et al. [9] and Khan et al. [10] are adopted. As shown in Fig. 14 (a), the cross sectional area of the present setup can be divided into three separate control volumes where CV1, CV2 and CV3 represent the control volumes for the heat sink, top bypass area and side bypass area, respectively. By applying force balance, the pressure drop across each control volume can be written as Eqs. (26) - (28). In addition, by assuming that the pressures of the control volumes are equal at points \( P_1 \) and \( P_4 \), as depicted in Fig. 14 (b), Eqs. (26) - (28) can be simplified to Eq. (29). Finally, by applying mass conservation [Eq. (30)] and eliminating \( U_2 \) and \( U_3 \), \( U_1 \) can be written in the form as shown in Eq. (31).

\[
P_1 + \frac{1}{2} \rho U_1^2 = P_4 + \frac{1}{2} \rho U_2^2 \left( 1 + n_f D \right) \tag{26}
\]

\[
P_1 + \frac{1}{2} \rho U_1^2 = P_4 + \frac{1}{2} \rho U_2^2 \left( 1 + f_2 \frac{L}{D_{h,2}} \right) \tag{27}
\]

\[
P_1 + \frac{1}{2} \rho U_1^2 = P_4 + \frac{1}{2} \rho U_3^2 \left( 1 + f_3 \frac{L}{D_{h,3}} \right) \tag{28}
\]

\[
U_2^2 \left( 1 + n_f D \right) = U_2^2 \left( 1 + f_2 \frac{L}{D_{h,2}} \right) = U_3^2 \left( 1 + f_3 \frac{L}{D_{h,3}} \right) \tag{29}
\]

\[
AU = A_1 U_1 + A_2 U_2 + A_3 U_3 \tag{30}
\]

\[
U_1 = \frac{AU}{A_2 \sqrt{1 + n_f D \frac{L}{D_{h,2}}} + A_3 \sqrt{1 + n_f D \frac{L}{D_{h,3}}} + A_1} \tag{31}
\]
In Eq. (31), $A_1$, $A_2$ and $A_3$ are the cross sectional areas of the respective control volumes and $D_{h,1}$, $D_{h,2}$ and $D_{h,3}$ are the corresponding hydraulic diameters. On the other hand, $n_f$ is the number of fins per row and $D$ is the drag coefficient of each NACA 0024 airfoil, which can be further broken down into $C_s$ (skin friction drag coefficient), $C_p$ (pressure drag coefficient) and $C_i$ (induced drag coefficient), expressed in Eq. (32). In addition, $f_2$ and $f_3$ are the friction coefficients of empty flow channels which can be approximated using the Colebrook correlation [34] for turbulent flow [Eqs. (33) and (34)], where $\varepsilon$ is taken to be 0.0005 m for a smooth channel wall.

$$D = C_s + C_p + C_i$$  \hspace{1cm} (32)

$$\frac{1}{\sqrt{f_2}} = 2 \log \left( \frac{\varepsilon}{3.7D_{h,2}} + \frac{2.51}{Re \sqrt{f_2}} \right)$$  \hspace{1cm} (33)

$$\frac{1}{\sqrt{f_3}} = 2 \log \left( \frac{\varepsilon}{3.7D_{h,3}} + \frac{2.51}{Re_3 \sqrt{f_2}} \right)$$  \hspace{1cm} (34)

To solve Eq. (31), $C_s$, $C_p$ and $C_i$ have to be defined. As the NACA 0024 airfoil has a relatively small chord-to-length ratio, it is reasonable to approximate $C_i$ to that of a flat plate. Hence, for turbulent flow, $C_i$ can be expressed as Eq. (35). The pressure drag, on the other hand, arises from the presence of the wake which affects the flow and pressure distribution on the airfoil. Therefore, the change in $\alpha$ which resulted in the changed wake characteristics would also have direct influence on $C_p$. Recently, in the lift force and wake measurements performed by Alam et al. [32] on a NACA 0012 airfoil for $\alpha$ ranging from 0° to 90°, $C_p$ was determined to be a sine function of $\alpha$ [Eq. (36)]. Even though Eq. (36) was originally developed for NACA 0012, the correlation should also provide an accurate approximation for the NACA 0024 profile as both airfoils are symmetrical and have small chord-to-length ratios. Finally, by employing the Lifting Line Theory [33] and assuming an elliptical lift distribution over span of the fin, $C_i$ can be written as Eq. (37). To obtain the lift coefficient ($C_l$) in Eq. (37), the Thin Airfoil Theory [33] is further applied and by including the effect of apparent $\alpha$ as a result of the induced drag, $C_i$ can be expressed as Eq. (38). In Eqs. (37) and (38), $AR$ denotes the aspect ratio of the NACA 0024 fin.

$$C_s = \frac{0.074}{Re_1^{0.2}}$$  \hspace{1cm} (35)

$$C_p = 1.35 \sin \alpha$$  \hspace{1cm} (36)
By substituting Eqs. (32) - (38) into Eq. (31), solutions for $U_1$ can be obtained. However, as Eqs. (33) and (34) have to be solved implicitly, an iterative scheme is required to determine $f_2$ and $f_3$. In addition, as Eqs. (33), (34) and (35) contain $Re_1$, $Re_2$ and $Re_3$ which are in turn related to $U_1$, $U_2$ and $U_3$, an iterative method is also necessary to compute Eq. (31). Hence, two closed loop iterative schemes were constructed where the value of $U_2$ was used as the initial guessed values for $U_1$, $U_2$ and $U_3$ and a stopping criteria of $|U_{1,i+1} - U_{1,i}| < 0.0001$ was prescribed. Using MATLAB, the values of $U_1$ were computed for the range of $\alpha$ and Re tested and the results were further curve-fitted as functions of $U$ and $\alpha$ by the non-linear regression method, where the final form is given by

$$U_1 = 0.93 \exp^{-2.53\alpha} U$$

From Eq. (39), it can be seen that $U_1$ varies linearly with $U$ but decreases exponentially as $\alpha$ increases.

At this juncture, it is therefore appropriate to introduce $C_r$ as an exponentially increasing function of $\alpha$ to account for the thermal enhancement as a result of the vortices generated. Hence Eq. (25) can be rewritten as

$$C_r = A \exp^{B \pi \alpha}$$

In addition, the relationship between $D_{h,1}$ and $D_h$ can be expressed as

$$D_{h,1} = \left( \frac{H}{C} \right) \left( \frac{B}{W} \right) \left( \frac{W + C}{B + H} \right) D_h$$

Lastly, by substituting Eqs. (39), (40) and (41) into Eq. (23), a semi-analytical correlation of Nu_b for the NACA 0024 heat sinks is obtained as

$$Nu_b = A \exp^{B \pi \alpha} \left[ \left( \frac{H}{C} \right) \left( \frac{B}{W} \right) \left( \frac{W + C}{B + H} \right) 0.93 \exp^{-2.53\alpha} \right]^{m} \text{Re}^{m} \text{Pr}^{0.33}$$

As shown in Eq. (42), the present model recovers the Reynolds number (Re) which is only related to the flow channel hydraulic diameter ($D_h$) and the air velocity in the flow channel ($U$). In addition, the effects of air bypass (or air velocity through the heat sink) and heat transfer enhancement due to vortex generation have also been incorporated with the introduction of two additional exponential terms.

By curve-fitting the present experimental data, constants $A$, $B$ and $m$ are determined to be 17.59, 0.44 and 0.45, respectively. It should be noted that, even though Eq. (42) shows that by changing the heat sink-to-flow channel dimensional ratios $\left( \frac{H}{C} \right)$, $\left( \frac{B}{W} \right)$ and $\left( \frac{W + C}{B + H} \right)$, $Nu_b$ can also be affected, the constants $A$, $B$ and $m$ obtained in the present correlation do not account for this effect as these dimensional ratios were not varied in the experiments. Finally, by substituting the values of constants $A$, $B$ and $m$ and the dimensions of heat sink and flow channel $B$, $C$, $H$ and $W$ into Eq. (42) and with further simplification, the final form of the correlation is given by

$$Nu_b = 10.653 \exp^{0.24\alpha} \text{Re}^{0.45} \text{Pr}^{0.33}$$

A comparison of $Nu_b$ values predicted by Eq. (43) and the data obtained from the present experiments is shown in Fig. 15. It can be observed that the model provides reasonably accurate predictions of the
NACA 0024 heat sink at various Re and α. For Re ≥ 6800, the maximum deviation in Nu between the correlation and experimental results is 6.9% whereas at Re = 3400, up to 15% deviation is observed. It should be noted that as the relationship for the pressure drag coefficient \((C_p)\), as shown in Eq. (36), was developed for large Re values, it may be less accurate when modeling lower Re range. Hence, this may have resulted in the larger discrepancies observed at Re = 3400.

Fig. 15 Comparison of correlation [Eq. (43)] against experimental results.

5. Conclusions

In this paper, novel airfoil heat sinks fabricated by SLM were experimentally investigated in a rectangular air flow channel with \(CL_t\) and \(CL_h\) of 2.0 and 1.55 and Re ranging from 3400 to 24000. The significant findings of the present investigations are summarized as follows:

- In comparison with the circular heat sink, highest enhancements in \(Nu_b\) of 29% and \(Nu_t\) of 34.8% were recorded for the NACA 0024 at α = 0° whereas for the rounded rectangular heat sink, highest enhancements in \(Nu_b\) of 34.7% and \(Nu_t\) of 20.8% were recorded.
- The overall performances of the heat sinks were evaluated by averaging the \(Nu_b\) values for the range of Re tested. On this basis, the enhancements in \(Nu_{b,ave}\) and \(Nu_{t,ave}\) of the NACA 0024 heat sink with α = 0° as compared to the circular heat sink are approximately 11.5% and 16.6%, respectively.
- The overall thermal performances of the NACA 0024 heat sinks were observed to increase with increasing α of the NACA 0024 heat sinks. Even though less significant differences in thermal performance was observed with the increase in α from 0° to 5°, noticeable shifts in the \(Nu_b\) and \(Nu_t\) curves were obtained with further increments in α to 10° and above. In comparison with the circular heat sink, the highest enhancements in \(Nu_{b,ave}\) of 19.7% and \(Nu_{t,ave}\) of 25.2% were achieved with the NACA 0024 heat sink with α = 20°.
- Based on the experimental results, it was suggested that the streamline design of the airfoil heat sinks improves the heat transfer by reducing the effects of air bypass and hence, allow more air flow through the fin array. On the other hand, the thermal performance of the NACA 0024 heat
sinks was further enhanced with the increase in $\alpha$ as a result of the formation of vortices which induced fluid mixing.

- Based on these proposed mechanisms, a semi-analytical model is develop to characterize the heat transfer performance of the NACA 0024 heat sinks for the range of $\alpha$ and Re tested where reasonably accurate predictions of $Nu_b$ were achieved.

- Heat sinks in the form of pin fin arrays are commonly employed for the cooling of electronic components where they are mounted on circuit boards with clearances around them. Due to the higher flow resistance of the heat sink, air tends to bypass the heat sink and flow through the clearances, hence, degrading the heat sink performance and increasing the operating temperature of the electronic components. In this regard, the experimental results and the correlations developed from the present work provide the data and predictive tools for employing airfoil heat sinks in the thermal management of integrated circuit boards. However, for accurate system design, the present data and predictive tools should be used in a flow channel with $CL_t$ and $CL_h$ of 2.0 and 1.55, respectively and Reynolds number from 3400 to 24000.

Acknowledgements

The authors would like to acknowledge the assistance of undergraduate student Ming Chong Lim for performing some of the experiments presented in this paper. Funding for the SLM facility by the National Research Foundation, Singapore, is also acknowledged.

References


