Thermal Performance Characteristics of Integrated Cooling Solutions Consisting of Multiple Miniature Fans

J Stafford$^{1,4,5}$, F Fortune$^2$ and D Newport$^3$

$^1$Bell Laboratories, Alcatel-Lucent, Dublin, Ireland
$^2$Institut Catholique des Arts et Métiers, Toulouse, France
$^3$Stokes Institute, University of Limerick, Ireland

E-mail: jason.stafford@alcatel-lucent.com

Abstract. Thermal performance characteristics are assessed for multiple miniature axial fans of 24.6 mm diameter that provide impingement cooling on a finned surface. Combined experimental and numerical analyses indicate that fans positioned adjacently in an array can influence heat transfer performance both positively and negatively by up to 35% compared to an equivalent single fan – heat sink unit operating standalone. However the level of thermal performance reductions, coupled with greater geometrical flexibility, makes the design approach a viable alternative to current single fan – heat sink units. Experimental measurements also suggest that for a fixed spacing, fan operating point is a sensitive criterion for ensuring optimal thermal performance over an equivalent single fan unit. Numerical simulations, modelled using experimental inputs, have provided an insight into the flow fields produced by the interaction between adjacent fans and the finned geometry. Fluid recirculation occurs beneath the fan hub of the centrally located fan in the array, with the adjacent fans on the periphery experiencing cross flow in the hub region. A novel experimental approach utilising infrared thermography has been developed to assess the validity of the numerical model. Indeed, the previously stated flow features were confirmed using this assessment, while limitations in the modelling assumptions have been outlined. Overall, the results provide recommendations in the design of fan cooled heat sinks utilising multiple axial fans for jet impingement and an understanding of the flow physics which occur within this compact cooling solution design.

1. Introduction

Strategic cooling of electronic systems has been greatly assisted by the implementation of axial fans at both system and component levels. As heat density increases, there is a demand for novel ways to extend air cooling and efficiently dissipate high thermal loads in confined spaces. Scaling of a combined axial fan and heat sink assembly is often considered to address higher thermal loads; however this approach can be limited by available space or require a costly redesign of the electronics layout to accommodate larger units. In this study, an assessment of an alternative forced cooling approach using multiple fans to impinge air on a single heat sink assembly is considered.

Integrating multiple miniature fans as part of a heat sink assembly may have a number of benefits over geometrically scaling a single large fan to meet thermal requirements. Firstly, the thermal designer has greater freedom in the footprint design of the cooling solution. Large single fan units are typically positioned on heat sinks with a footprint the same dimension as the fan housing (square dimensions). A multiple miniature fan arrangement could have a footprint design of various dimensions depending on the layout of the fans i.e. $1 \times 2$ or $1 \times 3$ fan array (rectangular), $2 \times 2$ fan array (square), etc. Secondly, the use of many small diameter fan designs over a single large fan diameter design can reduce acoustic emissions and may

---

$^4$ Corresponding author.
$^5$ Research was carried out during previous position at Stokes Institute, University of Limerick, Ireland.
mitigate the adverse effect of fan blockage on thermal and acoustic performances for densely packed electronics.

Lin and Chou [1] examined fan performance characteristics for a number of axial fans with diameters from 60 mm to 80 mm. A board positioned 10 mm above the fan inlet reduced flow rates by ~ 16% and 20%, and increased noise levels by less than 10% and 13% for each fan design. Although these differences are relatively moderate, there is a clear trend showing the advantages of implementing smaller diameter fans in such space restricted environments. Therefore, miniature fans could potentially operate in confined environments with limited degradation in performance. However, other performance limitations due to increased viscous forces are also introduced through miniaturization and subsequently operating at low Reynolds numbers. Quin and Grimes [2] have documented this Reynolds number effect for micro axial fans.

Previous literature on the thermal performance due to impingement by multiple axial fan arrays is limited; however the recent attention on normally impinging jet arrays can be somewhat analogous. Jeffers et al. [3] conducted an experimental analysis of an array of water jets impinging fluid onto flat and structured target surfaces. Velocity field images of impingement on the flat surface highlighted the fluidic interactions between adjacent jets which caused reductions of ~ 15% in thermal performance compared to a single jet. As the flow from fans is swirling upon exit due to the centrifugal forces in the jet [4,5], examination of features observed for swirling impinging jets may also be comparable. Nuntadusit et al. [6] compared conventional impinging jets with jets created by twisted-tape swirl generators. Comparisons were made through experimental observations of impingement patterns created on an oil-film surface and also quantitative thermal data using calibrated temperature measurements of a thermochromatic liquid crystal sheet. It was determined that thermal performance due to a swirling jet array was up to ~ 20% above a corresponding conventional impinging jet array.

Despite recent attention on jet arrays, and the potential design benefits previously outlined, the authors have yet to observe reports discussing thermal and fluidic behavior of an array of fans as part of an integrated cooling solution. This study aims to address this through the examination of two and three-unit arrays using 24.6 mm diameter axial fans and a low profile (10 mm) finned heat sink geometry. Bulk thermal measurements have been conducted to determine the effect of adjacent fans, and also compare thermal performance to a single fan – heat sink unit. Numerical simulations of the fan – heat sink arrangements have also been carried out to investigate the flow features within the heat sink due to the interactions between adjacent fans. These simulations have been assessed through the development of a novel experimental apparatus coupled with infrared measurements to provide full-field validation.

2. Experimentation

This section describes the experimental approaches for characterising the thermal performance of various multiple fan – heat sink arrangements. Initial design information is outlined primarily, including fan performance and heat sink design. This is followed by details on the bulk thermal measurements and testing facilities. Finally, the local thermal measurement and testing facility is discussed.

2.1. Fan performance and heat sink design

Non-dimensional fan performance characteristics for the axial fan considered in this study are presented in figure 1. Flow rate and pressure rise data was measured using standardised (BS848) facilities previously outlined by Stafford et al. [4]. A hub-tip ratio of 0.66 produces the high pressure coefficient ($\psi$) – low flow coefficient ($\phi$) characteristics observed in figure 1, with a sharp change in performance indicating the stalling point of the fan. Swirl angle of the fan outlet flow was also measured to be 39.8° for a nominal rotational speed ($\omega$) of 9000 rpm [4]. These fan performance attributes were used as inputs to the numerical models which will be discussed in a proceeding section.

A commercially available plate-fin aluminium heat sink ($k = 220\, \text{W/mK}$) was selected for the current study. This has fin spacing 2.7 mm, fin height 8.3 mm, and also has multiple cross-cuts which allow air to mix between channels. The dimension of the heat sink for each single axial fan is $35\, \text{mm} \times 35\, \text{mm}$. An image of a single fan and heat sink arrangement is shown in figure 2. Two different fan – heat sink designs were considered. The first design allows air to exit the heat sink from all sides (4-exit), whereas the second design contains only two exits for air to leave the heat sink (2-exit). The second design was considered as it is often necessary in electronic enclosure design to restrict the heated outlet air from heat sinks to exit vents, or a certain direction away from thermally sensitive components nearby.

It was necessary to assess fin efficiency to ensure the heat sink design could perform effectively for the current arrangements and flow configurations. Fin efficiency ($\eta$) was calculated using the solution to the one-dimensional differential equation for heat conduction within a fin which has a convective/radiative
boundary and an insulated fin tip. This solution is appropriate as the fin tip in the experiment is insulated by a clear polycarbonate confinement plate as seen in figure 2.

\[
\eta = \sqrt{\frac{R_c}{R_s}} \tanh \sqrt{\frac{R_c}{R_s}}
\]  

(1)

where \( R_s = 1/hA_s \) the combined convective and radiative resistance from the fin surface (\( h \) is heat transfer coefficient, \( A_s \) is fin surface area), and \( R_c = H/kA_c \) the conductive resistance from base to fin tip (\( H \) is fin height, \( k \) is fin thermal conductivity, \( A_c \) is fin cross-sectional area). For the range of scenarios considered, fin efficiency \( \eta > 97\% \).

![Figure 1. Non-dimensional fan performance curve.](image1)

![Figure 2. Single axial fan mounted to a) 4-exit and b) 2-exit heat sink designs.](image2)

2.2. Thermal resistance measurements

Figure 3 presents the experimental arrangement for bulk thermal measurements of a multiple fan solution. It was necessary to investigate the thermal performance provided by each individual fan within the array. Each array of fans impinged air on a heat sink which consisted of modules. One module within the heat sink was a heated aluminium section, whereas the other modules were manufactured from polycarbonate and unheated. Thermal resistance data was then calculated for each heated module in order to determine if thermal performance for individual fans within an integrated cooling solution differed to a single fan – heat sink unit operating in standalone.

A minco thin-film polyimide heater (OD 34.2 mm, ID 13.3 mm) was used as a heat source and attached to the base of the heat sink / module. A TTi Dual power supply was used to supply power to both the heater and fans. Power to the fans was supplied through additional circuitry consisting of variable resistors which allowed independent adjustment of fan speed for all fans within an array. An Omega HHT13 optical tachometer was used to monitor fan speeds which ranged from 4000 – 12000 rpm. Two K-type thermocouples, calibrated to 0.1°C, were embedded in the heat sink base and the top of the furthest fin from the heat source. During all experiments, temperature measurements for both points on the heat sink differed by a maximum of 5%, reinforcing that an almost isothermal boundary condition existed. A third calibrated K-type thermocouple was used to measure the ambient inlet air temperature to the fans. All thermocouples were connected to a National Instruments carrier and temperatures were recorded using LabVIEW 8.

Total thermal resistance was calculated as:

\[
R_T = \frac{T_w - T_a}{Q_m}
\]  

(2)

where \( Q_m \) is the measured power input to the heater, \( T_w \) is the average steady state temperature of the heated module and \( T_a \) is the ambient inlet air temperature. Each experiment was carried out at a temperature difference of \( T_w - T_a = 45^\circ C \). Eq. (2) represents the total heat dissipation through multiple heat transfer modes. Forced convection thermal resistance, \( R_c \), is of interest in the current study and was calculated based on a thermal network representation of the experimental configuration.
where $R_L$ is the thermal resistance due to passive losses, $R_{sp}$ is the resistance to heat spreading from the heater to the base, and $R_{cd}$ is the resistance produced by the interface material used to mount the heater to the base. In Eq. (3), $R_L$ was measured experimentally by setting the fan to 0 rpm, and packing the finned geometry exposed to the fan outlet flow during normal operation with insulation. Input heater power was then adjusted such that $T_u - T_w = 45^\circ \text{C}$.

2.3. Local thermal assessment
An approach developed to accurately characterize the local heat transfer performance due to fan flows [7] was implemented to examine the flow field within a multiple fan – finned heat sink arrangement and is provided in figure 4. It was decided that for validation purposes, the most complex flow arrangement would be examined and it was anticipated that the three fan – 2-exit heat sink arrangement would suffice.

A full polycarbonate prototype of the finned heat sink was manufactured without a base. The polycarbonate heat sink was then bonded to a tensioned stainless steel (SS304) thin foil with a measured thickness of 41.7µm. Once the bonding process was completed and all axial fans were mounted to the polycarbonate heat sink, the thin foil was tensioned between two copper bus bars and Joule heated using a TTI TSX1820P DC power supply resulting in an input heat flux of $1\text{ kW/m}^2$. Fan rotational speed was controlled as described in the previous section. An Indigo Merlin infrared camera with a 25 mm lens that provided an angular field of view of $16^\circ \times 22^\circ$ was used to record thermal maps of the underside of the heated-thin-foil which was painted matt black. An in situ calibration of the camera was conducted to reduce uncertainty in the measured temperature from 2.8$^\circ \text{C}$ to 0.2$^\circ \text{C}$. A description of this procedure has been documented elsewhere by Stafford [8]. Ambient inlet air temperature to the fans was monitored as described in section 2.2.

A temperature map of the heated base plate encompasses the effects of the impinging air flow from the array of fans. This temperature profile shall be used to assess the numerical model.

2.4. Uncertainty analysis
An uncertainty analysis was considered using the sequentially perturbed method of Moffat [9]. Uncertainty in measured variables voltage, current, temperature, and distance were estimated 0.01 V, 0.01 A, 0.1$^\circ \text{C}$, and 0.1 mm for bulk thermal measurements. Consequently, this resulted in an uncertainty of 3.77% for $R_{pg}$. Uncertainty in the local temperature measured using infrared thermography was 0.2$^\circ \text{C}$. The resolution of the optical tachometer which measured fan rotational speed is 1 rpm; however an uncertainty of 50 rpm was estimated due to variations monitored over the test duration.

3. Numerical Analysis
Numerical simulations have been conducted using ANSYS Fluent computational fluid dynamics software. A second order, double precision discretization scheme was chosen to solve for the conservation equations of momentum and energy. The primary purpose of these simulations was to assist in the discussion of the hypothesized flow fields within the current experimental arrangements. The simulations, coupled with the
previously discussed validation method, will also be used to highlight limitations of current modelling routines for virtual fan – heat sink design.

3.1. Numerical models
Numerical models were generated to encompass the range of cooling solutions experimentally examined. An example of one such model is provided in figure 5. Despite operating at low Reynolds numbers (Re$_{\text{a}} < 300$ based on fan flow rates and annular slot width of fan flow area as characteristic length scale, $d$) the flow regime was defined as turbulent based on the empirical information of Stafford et al. [4] who demonstrated that heat transfer scaled towards a turbulent regime. An enhanced RNG two-equation turbulence model was selected based on empirical observations of the base plate surface temperature distribution in the current study. The RNG model has also been found to produce realistic simulations for applications involving similar rotating flows [10].

The fan performance characteristics discussed in section 2.1 were used as inputs to a lumped fan model representing the axial fan. This was a three dimensional fan, as it was necessary to examine if interference between adjacent fan inlet airflows existed. It was concluded that the current fan spacing has a minor influence on the inlet flow. As a result, variation to performance could be solely attributed to the interaction of the adjacent outlet flows. The heat sink was modeled using full geometrical details. Air enters the domain to the inlet at ambient temperature and exits the heat sink to ambient pressure.

3.1.1. Grid independence. A grid independence study was conducted on the computational domain of the single fan – 2-exit heat sink example. It was determined that using a grid size of 501,130 nodes provided the best compromise between accuracy and computational demand, with calculated heat sink thermal resistance differing by $< 1\%$ compared to a grid size of $2.12 \times 10^6$ nodes. The grid independence study was limited to this single fan case and the resultant mesh parameters were then extended to the other models. This reduced the amount of trials and was feasible due to each model having similar fin layout and spacing.

3.2. Validation of numerical model
A numerical model was developed to represent the experimental apparatus in section 2.3. The thin foil was modeled as an isotropic 2-D wall with an effective thickness of 63.5µm, which is the cumulative thickness of both SS304 and matt paint layers. An effective thermal conductivity was then calculated as $k_{\text{eff}} = k_f \left( t_f / (t_f + t_p) \right) + k_p \left( t_p / (t_f + t_p) \right)$, equating to 11.182 W/m.$^\circ$C in the current study. These assumptions are valid based on a Biot number of $< 0.001$, and the experimental findings of Stafford et al. [7].

As the main focus is on the forced flow field created by the axial fans, both the gravity vector and radiation were excluded to reduce computational requirements. This was possible by inversely calculating the total heat dissipation due to forced convection from the experimental arrangement using an energy balance of the heated-thin-foil [7]. This heat rate was then applied as a boundary condition to the thin-foil in the proposed numerical model.

A comparison between measured and simulated temperature profiles is presented in figure 6 for a fan rotational speed of 9000 rpm. Fan outlet profiles have been superimposed to assist in the discussion. The experimental temperature profile in figure 6 a) indicates an increase in temperature beneath the hub region of the central fan. This is not apparent for the peripheral fans due to the crossing air flow produced by the central fan. This crossing air flow also reduces heat transfer performance in a region between central and peripheral fans within the array. Similar features are also evident in the numerical prediction of figure 6 b).

6th European Thermal Sciences Conference (Eurotherm 2012) IOP Publishing
Journal of Physics: Conference Series 395 (2012) 012029 doi:10.1088/1742-6596/395/1/012029
virtual design stages. However, it is clear that the outlet flow interactions between adjacent fans is a
dominant feature within the heat sink and is captured sufficiently by the simplified numerical approach.
Quantitatively, the numerical results demonstrate an under-prediction of the thermal performance of the
multiple fan arrangement. An issue which may be improved by implementing more detailed information on
the fan units as previously mentioned. Despite these modeling limitations, a qualitative validation ensures
that the simulations provide a suitable insight for discussing the flow fields within the multiple fan integrated
cooling solutions.

![Image](image-url)

**Figure 6.** Temperature distribution on the base of a three fan cooling solution using a) experimental
measurement and b) numerical simulation. ($\omega = 9000$ rpm, $T_c = 27^\circ$ C, Contour level: 0.7 $^\circ$C)

4. Results and discussion

In this section, thermal resistance measurements are presented followed by a discussion on the simulated
velocity fields for each of the experimental arrangements.

4.1. Thermal resistance

The thermal performance of single, two and three fan cooling solutions have been assessed experimentally.
This has been completed for two heat sink designs presented in figure 2; one which allows air to exit all
sides of the heat sink (4-exit) and a second which only allows the air to leave two sides (2-exit).

Figure 7 presents the forced convective thermal resistance measurements for the 4-exit design. A single
fan solution is compared with the multiple fan arrangements over the range of fan rotational speeds
considered. A fan within a two fan array provides thermal performance that is relatively unaffected by the
adjacent flow interactions up to a speed of 10,000 rpm. At this speed and above, thermal performance was
shown to reduce by as much as 25% compared to a single fan unit operating in standalone. The most
significant variation in performance is for the central fan (C) in a three fan array, where $R_c$ is reduced by
almost 35% at the nominal operating speed of 9000 rpm. However this improvement in thermal performance
reduces with increasing rotational speed to 13% at 12,000 rpm. Peripheral fans (P) of the three fan array
follow a similar trend as peripheral units in the two fan array, providing a 21% reduction in thermal
resistance at 7000 rpm then changing to an 11.5% increase in thermal resistance at 10,000 rpm. The findings
suggest that improvements can be achieved; however it appears to be dependent on balancing fan operation
with fan-to-fan distance within the array.

Thermal resistance measurements of the 2-exit design are provided in figure 8. In contrast to the previous
design, the use of multiple fans in this type of arrangement generally degrades thermal performance over a
single fan 2-exit solution. This has been attributed primarily to the exit flow area remaining fixed for all fan
arrangements. A fan within a two fan array demonstrated a reduction in thermal performance of $\sim 20$
throughout the range of speeds examined. The central fan within the three fan array showed a reduction of
30.5% at 9000 rpm. This fan must overcome a higher flow resistance than the peripheral units, as the only
path for air to leave the heat sink is by crossing the peripheral fans which have only minor differences in thermal performance compared to a single fan unit.

The overall forced convection thermal resistance at a nominal 9000 rpm for the three fan cooling solutions has been predicted as 0.674 °C/W (4-exit) and 1.06 °C/W (2-exit) using the individual data on central and peripheral modules within the heat sink. Consequently, low thermal resistance solutions can be achieved for space restricted environments through the use of multiple fans as part of an integrated cooling solution. In the current study, the total profile of the solution was less than 20 mm.

4.2. Simulated velocity fields

A number of sample velocity fields have been selected from the three-dimensional simulations to discuss the flow features which exist in the multiple fan solutions. These are presented in figure 9 – 11 for a rotational speed of 9000 rpm. YZ planes are positioned near (2.1 mm) the fan centre of rotation, coinciding with the centre of a heat sink channel.

Distinct differences in the flow field within a heat sink occur when moving from a single fan to a multiple fan arrangement. The 4-exit design maintains relatively consistent impingement zones for both peripheral and central fans when compared to a single fan scenario. A three-dimensional flow recirculation is observed where adjacent fan outlet flows meet. It is possible that the mixing which occurs between adjacent fans results in an improvement of the thermal performance noted in the previous section. Indeed, heat sink surfaces beneath the recirculation zone are constantly being replenished with cool air, which then exits the heat sink sides. Figures 10 a) and 11 a) also demonstrate the effect of these interactions on the angle of impingement. Impingement occurs almost normal to the base of the finned heat sink, and covers a larger area of the finned structures, including beneath the fan hub which is not evident in a single fan arrangement. At the exit of the heat sink, the angle of impingement reverts to that of a single fan arrangement. In this region, the air is diverted out of the heat sink directly after impingement due to low fluid resistance, and much of the convective surface area of the heat sink avoids interaction with this high velocity outlet flow. Hence, the recirculation zone benefits thermal performance, as it effectively interacts with the finned structures despite having a reduced velocity compared to the jet impingement near the heat sink outlet.

The numerical results presented for the 2-exit design in figures 10 b) and 11 b) indicate cross flow within the heat sink, similar to that observed in the literature for an array of normally impinging jets. Comparing the flow fields of the previous 4-exit design, it is clear that the occurrence of cross flow within the 2-exit heat sink diminishes the impingement velocity, and ultimately the thermal performance capability. This is particularly evident in the three fan arrangement, as the outlet flow from the central fan must pass the impingement zones of the peripheral fans to exit the heat sink. The central fan was found to operate below the point of stall at 9.4% of total fan pressure rise which reduced the flow rate by 13.3% compared to peripheral units. This ultimately reduces thermal performance capabilities for the central fan, confirmed by the thermal resistance data in figure 8.

Although there is a significant difference in the flow field of the peripheral fans for the 2-exit design compared to a single fan unit, thermal performance remained at a similar level. It is postulated that the increased momentum in the channels of the peripheral section of the heat sink (produced by the central fan) negates the adverse effect of cross flow on bulk thermal performance. This is reinforced by the thermal measurements of the peripheral region in a two fan array which showed 20% reductions in thermal
performance. Within the two fan arrangement, an increase in channel velocity is not apparent due to the absence of a central fan. Therefore, the disruption to the swirling jet and blockage by the adjacent fan has a detrimental influence on thermal performance.

**Figure 9.** Streamlines of a single fan (4-exit) cooling solution.

**Figure 10.** Simulated velocity fields for two fan array with a) 4-exit and b) 2-exit heat sinks.

**Figure 11.** Simulated velocity fields for three fan array with a) 4-exit and b) 2-exit heat sinks

5. Conclusions
A combined experimental and numerical study has provided an initial assessment of the capabilities of using multiple fan – heat sink cooling solutions for space restricted applications. Improvements and reductions in heat transfer performance were observed and attributed to the complex flow interactions within the heat sink, including fluid recirculation and cross flow. Bulk heat transfer enhancements of 35% were achieved when utilizing a multiple fan array with an open-sided heat sink design. However, similar magnitude reductions in heat transfer are experienced when the exit flow is confined to just two outlets from the heat sink. It has been inferred from numerical simulations that this degradation is due to the central fan causing a substantial cross flow effect on the neighboring fan impingement zones. Increased fan pressure rise also occurs as the swirling jet of air produced by the central fan must oppose that of the two adjacent fans to exit the heat sink geometry. The results suggest that although reductions in performance can occur, it can be outweighed by the design flexibility when trying to meet strict profile height and footprint dimensions.

**References**

[1] Lin S C and Chou C A 2004 *Appl. Therm. Eng.* **24** 2375-89
[2] Quin D and Grimes R 2008 *J. Fluids Eng.* **130** 101101 (1-10)
[3] Jeffers N M R, Punch J, Walsh E J and McLean M 2010 *J. Th. Sci. Eng. App.* **2** 041004 (1-11)
[4] Stafford J, Walsh E and Egan V 2010 *Int. J. Heat and Fluid Flow* **31**(5) 552-560
[5] Stafford J, Walsh E, Egan V and Grimes R 2010 *Int. J. Heat Mass Trans.* **53**(25-26) 5629-38
[6] Nuntadusit C, Wae-hayee M, Bunyajitradulya A and Eiamsa-ard S 2012 *Int. Comm. Heat Mass Trans.* **39** 102-7
[7] Stafford J, Walsh E and Egan V 2009 *Meas. Sci. and Tech.* **20**(10) 105401 (1-11)
[8] Stafford J 2010 On the miniaturisation of convection cooling solutions applicable to portable electronic devices Ph.D Thesis University of Limerick Ireland
[9] Moffat J R 1997 *Thermal Measurements in Electronics Cooling: Uncertainty Analysis* (Boca Raton FL: CRC Press) pp. 45-80.
[10] Shankaran G V and Dogruoz M B 2010 Validation of an advanced fan model with multiple reference frame approach *12th IEEE ITherm* 2-5 June Las Vegas NV