ON THE MINIATURISATION OF CONVECTION
COOLING SOLUTIONS APPLICABLE TO
PORTABLE ELECTRONIC DEVICES

Jason Stafford, BEng(Hons) Mech. Eng.

A thesis submitted for the degree of Doctor of Philosophy
at the Faculty of Science and Engineering,
University of Limerick, Ireland

Supervisors
Dr. Edmond Walsh and Dr. Vanessa Egan
Stokes Institute
Department of Mechanical and Aeronautical Engineering
University of Limerick

Submitted to the University of Limerick, November 2010
Declaration
The substance of this thesis is the original work of the author, and due reference and acknowledgement has been made, where necessary, to the work of others. No part of this thesis has been submitted in canditure for any degree.

Jason Stafford (Candidate)

Dr. Edmond Walsh (Supervisor)

Dr. Vanessa Egan (Supervisor)

This thesis was defended on the 11th of November, 2010.

Examination Committee
Chairman Dr. John Jarvis University of Limerick
External Examiner Dr. Amy S. Fleischer Villanova University
Internal Examiner Dr. Tara Dalton University of Limerick
Abstract

Dimensional restrictions in electronic equipment have resulted in miniaturisation of many existing cooling technologies. In addition, cooling solutions are required to dissipate increased thermal loads to maintain component reliability and user comfort. Fans are widely used in electronics cooling to meet such thermal demands, either in standalone for direct component cooling, or in combination with a heat sink. The thermal performance of such designs when scaled to dimensions suitable for use in portable electronics has received limited attention, mainly due to the reliance on passive cooling methodologies currently employed. However, as heat flux increases, passive cooling is reaching its limit and other solutions will be required. This thesis aims to address this issue by experimentally examining the fluid dynamics and thermal performance of forced convection cooling solutions with dimensional constraints.

Conventional finned and novel finless heat sink designs have been integrated with commercially available radial blowers to investigate cooling solutions with overall footprint areas as low as 487\( \text{mm}^2 \), and profile heights less than 5mm. The novel finless geometry, with reduced manufacturing cost, energy consumption and weight promoted heat transfer above that of the same size classical finned designs for a range of operating points. Both geometries showed increases of up to 20\% in thermal performance by aligning the fan exit flow with the heat sink channels, hence demonstrating the need for integrated fan and heat sink design of low profile applications. Optimisation and geometry selection criteria were determined by scaling profile height for both heat sink designs from 4mm to 1mm. Theoretical predictions underestimated the finless design thermal performance, which was found to scale towards that of a turbulent flow regime despite the low Reynolds number. The mechanisms of this improvement in heat flux was investigated and unique, heat transfer enhancing, features in the finless design were identified.

A combined infrared thermography and heated-thin-foil technique was developed for miniature fan applications, to accurately determine local heat transfer coefficients due to radial and axial fan flows. This highlighted the non-uniform heat transfer rates produced by the three-dimensional air patterns from rotating fans, and has been shown to be an important consideration in the design stages for component cooling. For the same chip temperature, strategic positioning of electronic components resulted in up to three-fold gains in power dissipation for direct component cooling applications. Local peaks in heat transfer coefficient when using axial fan impingement were directly related to the air flow and fan motor support interaction. It was found that for optimum thermal performance, motor support dimensions should be kept to a minimum and positioned on the inlet flow plane, the opposite to the current industrial practice.

Flow structures and surface heat transfer trends due to radial fan flows were found to be common over a wide range of fan aspect ratios (blade height to fan diameter). The limiting aspect ratio for heat transfer enhancement was 0.3, as larger aspect ratios were shown to result in a reduction in overall thermal performance. Results also indicate that low profile radial fan designs are not just limited to portable devices, but may also be a practical solution to thermal management issues in larger scale electronics. A practical operating condition was represented by the introduction of a uniform crossing air flow above a radial fan inlet and indirectly reduced surface heat transfer. A distorted inflow shifted the surface heat transfer distribution from an axisymmetric to asymmetric profile for a radially discharging fan design. Overall, the findings presented in this thesis are fundamentally and practically useful for the design of forced convection cooling solutions using rotating fans in space restricted applications.
Acknowledgements

Firstly, I would like to express my gratitude to my supervisors, Dr. Edmond Walsh and Dr. Vanessa Egan, for giving me the opportunity to undertake this research and allowing me to work under my own initiative. Their support, guidance and encouragement was unwavered over the duration of this research, and much appreciated. Professor Yuri Muzychka deserves thanks for his advice and assistance during his visit to Stokes Institute.

I would like to thank all the research staff at Stokes Institute who have helped me along the way to completing this work. The support of the technical staff of Paddy O’Regan and David Jones, and the administrative staff of Fionnuala O’Connell and Caroline Rafter, also deserve considerable praise. I must also thank the research students, in particular those I had the pleasure of sharing an office with, Stefano Chiavaroli, Oliver Taheny, Paul Fleming, Fiachra O’Leary, Niall O’Keeffe, Daithi Moore, Thomas Confrey, David Kinahan, Angela Morris, Mark Dalton and James Howard.

I would like to thank my friends David, Stephen, Ciaran, Richard, Ted, Maurice, Joe and Philip, for the many entertaining discussions over the past three years.

I would like to thank my family and in particular my parents, Lorcan and Mary, and also my brother Mark, for their invaluable advice and support throughout my life.

Finally, a special thanks to Theresa, whose support, love and inspiration made this challenging journey manageable.
To Lorcan and Mary, for their support and encouragement
# Contents

Abstract i

Acknowledgements ii

List of Tables viii

List of Figures xx

Nomenclature xxi

1 Introduction 1
  1.1 Motivations ......................................................... 1
  1.2 Literature .......................................................... 6
    1.2.1 Thermal management ......................................... 6
    1.2.2 Fan designs ................................................ 10
  1.3 Objectives .......................................................... 20
  1.4 Thesis structure .................................................. 21

2 Heat exchanger theory 22
  2.1 Fluid flow configuration .......................................... 22
  2.2 Heat exchanger model ............................................. 26
  2.3 Closure .............................................................. 29

3 Experimental techniques and facilities 30
  3.1 Mean heat transfer measurements ................................. 30
    3.1.1 Miniature cooling solutions ................................. 31
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1.2</td>
<td>Low profile cooling solutions</td>
<td>36</td>
</tr>
<tr>
<td>3.1.3</td>
<td>Component placement</td>
<td>39</td>
</tr>
<tr>
<td>3.2</td>
<td>Characterisation of fan performance</td>
<td>41</td>
</tr>
<tr>
<td>3.2.1</td>
<td>Experimental setup and procedure</td>
<td>42</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Data analysis</td>
<td>43</td>
</tr>
<tr>
<td>3.3</td>
<td>Uncertainty analysis</td>
<td>44</td>
</tr>
<tr>
<td>3.4</td>
<td>Closure</td>
<td>45</td>
</tr>
</tbody>
</table>

| 4        | Advanced experimental techniques                                      | 46   |
| 4.1     | Local heat transfer measurements                                      | 46   |
| 4.1.1   | Heated-thin-foil technique                                            | 47   |
| 4.1.2   | Assessment of the heated-thin-foil technique                          | 51   |
| 4.1.3   | Infrared thermography and camera calibration                          | 54   |
| 4.1.4   | Experimental setup and procedure                                      | 55   |
| 4.2     | Validation of heated-thin-foil correction method                      | 61   |
| 4.2.1   | Refinement of natural convection corrections                          | 70   |
| 4.2.2   | Turbulence detection and frequency response                           | 74   |
| 4.3     | Velocity field measurements                                           | 80   |
| 4.3.1   | Particle Image Velocimetry                                           | 80   |
| 4.3.2   | PIV system                                                            | 81   |
| 4.3.3   | Experimental setup and procedure                                      | 82   |
| 4.3.4   | Image processing                                                      | 85   |
| 4.3.5   | Data analysis                                                         | 86   |
| 4.3.6   | Statistical analysis                                                  | 87   |
| 4.4     | IR and PIV measurement planes                                         | 92   |
| 4.5     | Crossflow generator characteristics                                   | 93   |
| 4.6     | Uncertainty analysis                                                 | 99   |
| 4.7     | Closure                                                              | 101  |

| 5        | Miniature and low profile cooling solutions                           | 102  |
| 5.1     | Miniature cooling solutions                                           | 103  |
5.1.1 Flow alignment diffuser ........................................ 104
5.1.2 Thermal performance ........................................... 110
5.2 Low profile cooling solutions ..................................... 114
  5.2.1 Pressure and flow rate characteristics .................... 114
  5.2.2 Thermal performance .......................................... 117
5.3 Direct component cooling .......................................... 125
  5.3.1 Component placement ......................................... 126
  5.3.2 Flush and raised component orientation .................. 130
5.4 Closure .................................................................. 132

6 Heat transfer with Axial fans ........................................ 134
  6.1 Axial fan jet impingement ....................................... 134
    6.1.1 Pressure and flow rate characteristics ................ 135
    6.1.2 Local surface heat transfer ............................... 137
    6.1.3 Component placement ..................................... 144
    6.1.4 Exit flow characteristics ................................ 146
    6.1.5 Radial heat transfer ....................................... 150
  6.2 Motor support effects ........................................... 153
  6.3 Closure .................................................................. 155

7 Heat transfer with Radial fans ....................................... 157
  7.1 Radial fans and finless heat sink .............................. 158
    7.1.1 Pressure and flow rate characteristics ................ 159
    7.1.2 Velocity field ............................................... 161
    7.1.3 Local and average surface heat transfer .............. 170
    7.1.4 Summary ...................................................... 188
  7.2 Radial fans and finless heat sink with a crossing air flow .. 192
    7.2.1 Velocity field ............................................... 193
    7.2.2 Local and average surface heat transfer .............. 204
  7.3 Closure .................................................................. 219
8 Conclusions and Recommendations

8.1 Conclusions ................................................................. 221
8.2 Recommendations ...................................................... 226

References ................................................................. 228

A Publications .............................................................. A-1

B Experimental Appendix .................................................. B-1

B.1 Calibrations ................................................................. B-1
  B.1.1 Temperature recording instrumentation ...................... B-1
  B.1.2 Fan characteristic rig ............................................ B-2
  B.1.3 Infrared camera (Temperature) ............................... B-2
  B.1.4 Infrared camera (NUC) ........................................ B-7

B.2 Assessment of a heated-thin-foil top plate ....................... B-9

B.3 Fan motor and shaft support housing ............................ B-10

B.4 Sample uncertainty calculation ................................. B-11
## List of Tables

3.1 Experimental test conditions for component placement analysis. 40

4.1 Foil and paint specifications. 53
4.2 Axial fan specifications. 56
4.3 Radial fan specifications. 59
4.4 PIV system components for each experimental apparatus. 81
4.5 PIV system parameters for each experimental apparatus. 84
4.6 Crossflow velocities. 99

6.1 Jet and swirl angles of exit flow. 148
List of Figures

1.1 The personal computer industry: past, present and future trends, replotted from Juliussen and Feegan (2010). .......................................................... 3
1.2 Fluid structures a) and vorticity b) generated by a miniature radial fan operating at $Re = 115$. The experimental configuration is detailed in c). ......... 19

2.1 A typical radial fan and heat sink arrangement for a) finned and b) finless designs. .......................................................... 23
2.2 Flow configuration along heat sink channels in both finned and finless designs 23
2.3 Layout of a a) multiple fluid heat exchanger and a b) single fluid isothermal wall heat exchanger. .......................................................... 27

3.1 Schematic of a) finned and b) finless heat sinks outlining the overall dimensions and the optimised parameters $s$ and $t_{fin}$. .......................... 32
3.2 Velocity triangle showing fan exit angle. .......................... 33
3.3 Finned (top) and finless (bottom) cooling solutions employing straightening diffusers at the fan exit. .......................... 34
3.4 Schematic of a) finned and b) finless heat sink designs showing the common dimensions along with the varied parameters $H$ and $L$. Dimensions in millimeters (mm). .......................... 36
3.5 Finned (left) and finless (right) heat sink designs. Details of the fan outlet diffuser are also shown. .......................... 38
3.6 Finned (top) and finless (bottom) heat sinks showing reductions in profile height. .......................... 38
3.7 Experimental apparatus for component positioning on a radial finless heat sink. 40
3.8 Component positioning on polycarbonate plates. 41
3.9 Schematic of experimental flow rate and pressure rise test facility. 42
4.1 Heat transfer modes for heated foil and paint layers. 48
4.2 Ø24.6mm axial flow fan. 52
4.3 Ø48.5mm axial flow fan. 52
4.4 Cross-sectional images of the nominal a) 12.5μm thick foil and high emissivity paint, and b) 50μm thickness foil without paint. 53
4.5 Camera calibration fit with measured surface temperature data. 54
4.6 Error in infrared temperature measurement before and after in situ calibration. 55
4.7 IR experimental setup for radial fan designs and local heat transfer analyses for a) base plate, and b) top plate of a finless heat sink geometry. 58
4.8 Radial fan designs of a) 15mm, b) 24mm, c) 32mm, and d) 59mm diameters with fan profile height of 4mm. 59
4.9 Distinguishing between the crossflow and radial fan contributions to the top plate surface heat transfer distribution. 60
4.10 The transmissivity of polyethylene for various material thicknesses and radiant source temperatures, replotted from Tsilingiris (2003). 61
4.11 Temperature profile of a) the 14.3μm foil and b) the 41.7μm foil using the Ø24.6mm axial fan. c) Temperature difference between a) and b) for identical fan speeds. 62
4.12 Forced convection heat transfer coefficient a) using the 14.3μm foil and neglecting conductional effects; b) using the 41.7μm foil and neglecting conductional effects; c) error between a) and b); d) using the 14.3μm foil and accounting for all losses - Eq. (4.12); e) using the 41.7μm foil and accounting for all losses - Eq. (4.12); and f) error between d) and e). 64
4.13 Tangential conduction contribution using the Ø24.6mm axial fan with a) 14.3μm foil and b) 41.7μm foil. 65
4.14 Forced convection heat transfer coefficient using the Ø48.5mm axial fan and 14.3µm foil. ........................................... 66
4.15 Mean radial heat transfer coefficient using the Ø24.6mm fan. .......... 67
4.16 Mean radial heat transfer coefficient using a) nominal foil thickness data provided and neglecting paint coating, and b) using measured foil and paint thicknesses. ........................................... 68
4.17 Contribution of tangential conduction in the heated-thin-foil technique. .. 69
4.18 Natural convection streamlines above a heated horizontal surface facing upwards. ........................................... 71
4.19 Velocity magnitude and vectors over half of the plate width due to a heated horizontal surface facing upwards. ........................................... 71
4.20 A profile of the surface heat transfer coefficient on the camera viewing side of a heated horizontally orientated plate facing upwards. .......... 72
4.21 Scaling of Nusselt number with Rayleigh number for a horizontally orientated heated surface a) facing upwards and b) facing downwards. .......... 73
4.22 Mean Nusselt number for a horizontally orientated heated surface a) facing upwards and b) facing downwards. ........................................... 74
4.23 Fluctuations in the forced convection heat transfer coefficient using the Ø48.5mm fan with a) 14.3µm foil, and b) 41.7µm foil. .......... 75
4.24 Fluctuations in the surface temperature distribution using a) 14.3µm and b) 41.7µm thickness foils. ........................................... 76
4.25 Frequency and amplitude of a) the fluctuating forced convection heat transfer coefficient and b) the resultant thin foil temperature response for the case of a 14.3µm SS304 foil and 21.8µm paint coating. .......... 78
4.26 A non-dimensional representation of the influence of fluctuating heat transfer frequency on the resultant temperature amplitude. ........................................... 79
4.27 Maximum fluctuating frequency detectible using the heated-thin-foil and infrared thermography technique. ........................................... 79
4.28 PIV setup showing camera and laser positions with respect to the cooling solution arrangement. ........................................... 82
4.29 PIV setup for the analysis of the velocity field within a finless heat sink configuration. ................................................................. 83
4.30 The a) instantaneous velocity magnitude, b) time-averaged velocity magnitude, and c) Reynolds shear stress for a 1000 sample size. ............. 88
4.31 Velocity magnitude and Reynolds shear stress for: a),b) 75 samples; and c),d) 500 samples. Also included is percentage error, $E_R$, in velocity magnitude and Reynolds shear stress over a 1000 sample data set for: e),f) 75 samples; and g),h) 500 samples. ................................. 89
4.32 Probability density functions for velocity magnitude. ......................... 91
4.33 Stacked probability density functions indicating 95% convergence criteria for velocity magnitude, turbulence intensity, and Reynolds stress data. ... 91
4.34 IR and PIV measurement planes for the radial fan and finless heat sink experimental apparatus. ....................................................... 93
4.35 Combined radial fan and crossflow generator experimental apparatus. ... 94
4.36 PIV measurement planes A - D and IR measurement region E for characterisation of a crossflow generator. ................................. 95
4.37 Velocity magnitude and turbulence intensity in x- and z-directions for plane B without a confinement plate. ........................................ 96
4.38 Uniformity of the velocity magnitude and velocity vectors for plane D without a confinement plate. ................................................. 97
4.39 The local heat transfer coefficient a), fluctuations in the local heat transfer coefficient b), and average local Nusselt number in the streamwise direction c) for plane E without a confinement plate. ................................. 98
5.1 Normalised velocity magnitude data replotted from Egan et al. (2009), of flow in the entrance region of a) finned heat sink channels 1–6 and b) finless heat sink with no diffuser. All velocities are normalised with respect to the maximum velocity. ......................................................... 105
5.2 Normalised velocity magnitude plots depicting flow within the diffuser and heat sink entrance for both finned (left) and finless (right) geometries with diffuser angles of 25-60°. All velocities are normalised with respect to the maximum velocity. Regions in gray are shadows from the heat sink fins, which result from the laser light entering from the right. 106

5.3 Velocity magnitude plots of flow in a finned heat sink and 50° straight diffuser for fan speeds of a) 3000 rpm, b) 5500 rpm, and c) 8000 rpm. Respective scale bars for velocities in m/s are given above each result. Flow is from left to right. Regions in gray are shadows from the heat sink fins, which result from the laser light entering from the right. 108

5.4 Velocity magnitude plots depicting the flow within a finless heat sink and 25° diffuser for fan speeds of a) 3000 rpm, b) 5500 rpm, and c) 8000 rpm. Flow is from left to right. The dashed line represents the entrance to the heat sink from the diffuser. 109

5.5 Forced convection thermal resistance with varying fan rotational speed. Also indicated are the effects due to secondary cooling mechanisms (losses). 111

5.6 Forced convection heat transfer coefficient for both finned and finless heat sinks. 113

5.7 Fan performance curve at fan speed of 7000 rpm. 115

5.8 Fan and heat sink system performance curves. 116

5.9 Normalised finned heat sink mass flow rates showing predictions based on measured and manufacturers fan performance data. 117

5.10 Finned and finless heat sink thermal resistances for a) 4mm, b) 2mm, and c) 1mm profiles. 118

5.11 Forced convection Nusselt number, based on the inlet temperature difference, in both the entrance and fully developed region of an isothermal heat sink. 121

5.12 Ratio of heat transfer rates for heat sinks with a) 4mm, b) 2mm, and c) 1mm profiles. 122
5.13 Time-averaged velocity magnitude and streamlines between top and base plates for a Ø15mm fan with rotational speeds a) 4300 rpm and b) 7500 rpm.

5.14 Time-averaged velocity magnitude and streamlines between top and base plates for a Ø24mm fan with rotational speeds a) 4300 rpm and b) 7500 rpm.

5.15 Component heat transfer coefficient for varying positions, $X_o$, for a) a Ø15mm fan and b) a Ø24mm fan.

5.16 Component heat transfer coefficient and relative enhancement for flush and raised orientations.

5.17 The hypothesised air flow interaction for a raised component positioned on the base plate (section view).

6.1 Fan and system characteristics due to the inclusion of a flat plate downstream for a) Ø24.6mm and b) Ø48.5mm axial fans.

6.2 Forced convection heat transfer coefficient for a Ø24.6mm axial fan at 9000 rpm (left, a)-c)) and a Ø48.5mm axial fan at 4000 rpm (right, d)-f)) and various fan to plate distances. Ø24.6mm fan to plate distances: a) 5mm ($H/D = 0.203$), b) 10mm ($H/D = 0.407$), and c) 15mm ($H/D = 0.610$). Ø48.5mm fan to plate distances: d) 5mm ($H/D = 0.103$), e) 10mm ($H/D = 0.206$), and f) 15mm ($H/D = 0.309$). Contour level: 5W/m²K.

6.3 Section view of an axial fan above a flat plate with hypothesised interaction of the absolute velocity field with a motor support.

6.4 Local a) standard deviation, b) maximum, and c) minimum heat transfer coefficient fluctuations normalised with the time-averaged local heat transfer coefficient at 4000 rpm and $H/D = 0.309$. Contour levels: a) 0.05; b), c) 0.2.

6.5 Component locations 0 to 6 for $H/D = 0.203$ and 9000 rpm.

6.6 Mean heat transfer coefficient for each component with dimensions 5mm×5mm and fan speed of 9000 rpm.

6.7 Axial jet produced at a fan rotational speed of 9000 rpm.

6.8 Intersection of a diverging, helical-shaped fan outlet flow with flat surfaces downstream of an axial fan.
6.9 Helical trace of peaks in heat transfer coefficient with increasing \( H/D \) for all fan speeds examined. ........................................... 150
6.10 Mean radial distribution of Nusselt number for 9000 rpm and \( 0.203 \leq H/D \leq 0.610 \). ........................................... 151
6.11 Mean radial distribution of Nusselt number for the range of rotational speeds and at \( H/D = 0.203 \). ........................................... 152
6.12 Nusselt and Reynolds number scaling relationship for a) Ø24.6mm axial fan at \( H/D = 0.203 \), and b) Ø48.5mm axial fan at \( H/D = 0.309 \). ........ 152
6.13 Forced convection heat transfer coefficient for a) motor supports on outlet plane, b) motor supports on inlet plane, and c) no motor supports at 9000 rpm and \( H = 5\)mm. d) Mean radial distribution of heat transfer coefficient for a) - c). ........................................... 154

7.1 Non-dimensional pressure and flow coefficient for all radial fan diameters. .. 159
7.2 The trend in maximum a) flow rate and b) static pressure for a range of fan speeds and fan profiles with Ø15mm. ................................. 160
7.3 The contribution of Reynolds number to the maximum non-dimensional flow a) and pressure b) coefficients. ..................... 161
7.4 Time-averaged velocity magnitude and streamlines using a Ø15mm fan with \( H_f = 4\)mm \((a_r = 0.267)\) operating at a) 1000 rpm, b) 5000 rpm, and c) 10,000 rpm. d) Normalised velocity profiles of c) for a series of radial locations from the fan blades. ........................................... 163
7.5 Velocity field statistics within a finless heat sink design using a Ø15mm fan with \( H_f = 4\)mm operating at 10,000 rpm. a), b) Instantaneous velocity magnitudes at 0s and +1s; c) Turbulence intensity in the radial direction; d) Turbulence intensity in the axial direction; and e) Reynolds shear stress in the radial-axial plane. ........................................... 165
7.6 Time-averaged a) velocity magnitude and streamlines and b) normalised velocity profiles using a Ø32mm fan with \( H_f = 4\)mm \((a_r = 0.125)\) operating at 1000 rpm. ................................. 166
7.7 Time-averaged a) velocity magnitude and streamlines and b) normalised velocity profiles using a Ø59mm fan with $H_f = 4\text{mm}$ ($a_r = 0.068$) operating at 1000 rpm.

7.8 Velocity field statistics within a finless heat sink design using a Ø59mm fan with $H_f = 4\text{mm}$ operating at 1000 rpm. a), b) Instantaneous velocity magnitudes at 0s and +1s; c) Turbulence intensity in the radial direction; d) Turbulence intensity in the axial direction; and e) Reynolds shear stress.

7.9 Forced convection heat transfer coefficient (left) and fluctuations in $h_{fc}$ (right) on the base plate of a finless heat sink for a Ø15mm radial fan at 10,000 rpm and profile height of a), b) 2mm; c), d) 4mm; and e), f) 6.5mm. Contour levels: a), c), e) 5W/m$^2$K; b), d) 0.005; f) 0.1.

7.10 Forced convection heat transfer coefficient (left) and fluctuations in $h_{fc}$ (right) on the top plate of a finless heat sink for a Ø15mm radial fan at 10,000 rpm and profile height of a), b) 2mm; c), d) 4mm; and e), f) 6.5mm. Contour levels: a), c), e) 5W/m$^2$K; b), d) 0.01; f) 0.02.

7.11 Mean radial distribution of Nusselt number for varying fan profile height and 10,000 rpm for a) base and b) top plates.

7.12 Nusselt and Reynolds number scaling relationship on a) base and b) top plates using a Ø15mm radial fan with $H_f = 2\text{mm}$.

7.13 Defining the Richardson number based on the experimental arrangement.

7.14 Area average Nusselt number for varying fan profile height at 10,000 rpm.

7.15 Forced convection heat transfer coefficient on base (left) and top (right) plates of a finless heat sink for a radial fan with $H_f = 4\text{mm}$ and a), b) Ø24mm at 6000 rpm; c), d) Ø32mm at 1000 rpm; and e), f) Ø59mm at 1000 rpm. Contour levels: a), b) 5W/m$^2$K; c) - f) 3W/m$^2$K.

7.16 Normalised standard deviation (left) and minimum (right) fluctuations in $h_{fc}$ on the base plate of a finless heat sink for a radial fan with $H_f = 4\text{mm}$ and a), b) Ø24mm at 6000 rpm; c), d) Ø32mm at 1000 rpm; and e), f) Ø59mm at 1000 rpm. Contour levels: a), c) 0.01; b), f) 0.03; c) 0.02; d) 0.05.
7.17 Nusselt and Reynolds number scaling relationship on base (left) and top (right) plates using a), b) Ø24mm; c), d) Ø32mm; and e), f) Ø59mm radial fans with $H_f = 4\text{mm}$. .................................................. 184
7.18 Area average heat transfer coefficient for both axial and radial fans. .... 186
7.19 The influence of fan inlet confinement on the local heat transfer coefficient for a) $D/H_{CF} = 2.2$ and b) $D/H_{CF} = 8.8$. ................................. 188
7.20 Summary of the velocity field and surface heat transfer distributions using a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm. ......... 190
7.21 Summary of the velocity field and surface heat transfer distributions using a Ø59mm fan with $H_f = 4\text{mm}$ operating at 1000 rpm. ............. 191
7.22 Time-averaged velocity magnitude and streamlines using a Ø15mm fan with $H_f = 4\text{mm}$ operating at a) 1000 rpm, b) 5000 rpm, and c) 10,000 rpm under the influence of a crossing air flow of $\bar{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Region V1 presented. ................................. 194
7.23 Velocity field statistics within a finless heat sink design using a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm and under the influence of a crossing air flow of $\bar{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. a), b) Instantaneous velocity magnitudes at 0s and +1s; c) Turbulence intensity in the radial direction; d) Turbulence intensity in the axial direction; and e) Reynolds shear stress. Region V1 presented. ................................. 195
7.24 Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm and under the influence of a crossing air flow of $\bar{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Region V2 presented. ................................. 196
7.25 Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm and under the influence of a crossing air flow of $\bar{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Region V3 presented. ................................. 197
7.26 Time-averaged velocity magnitude and streamlines a) without a cross flow, and b) with a cross flow of $\overline{U}_{CF} = 2.06\text{m/s}$, in the vicinity above the inlet (V5) of a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm. 

7.27 Surface shear stress pattern for inlet flows with a cross flow, replotted from Holdo et al. (2000).

7.28 Velocity field between base and top plates (V1), and above the fan inlet (V4), for a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm. A fan inlet confinement of $H_{CF} = 6.7\text{mm}$ is presented a) without a cross flow and b) with a cross flow of $\overline{U}_{CF} = 1.48\text{m/s}$.

7.29 Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø59mm fan with $H_f = 4\text{mm}$ operating at 1000 rpm and under the influence of a crossing air flow of $\overline{U}_{CF} = 1.96\text{m/s}$ and $H_{CF} = 26.8\text{mm}$. Region V1 presented.

7.30 Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø59mm fan with $H_f = 4\text{mm}$ operating at 1000 rpm and under the influence of a crossing air flow of $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Region V1 presented.

7.31 Velocity field between base and top plates (V1), and above the fan inlet (V4), for a Ø59mm fan with $H_f = 4\text{mm}$ operating at 1000 rpm. Fan inlet confinement and cross flow velocity of a) $H_{CF} = 26.8\text{mm}$ and $\overline{U}_{CF} = 1.96\text{m/s}$, and b) $H_{CF} = 6.7\text{mm}$ and $\overline{U}_{CF} = 1.48\text{m/s}$ are presented.

7.32 Forced convection heat transfer coefficient on base (left) and top (right) plates of a finless heat sink for a Ø15mm radial fan with $H_f = 4\text{mm}$ operating at a), b) 1000 rpm, c), d) 5000 rpm, and e), f) 10,000 rpm under the influence of a crossing air flow of $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Contour level: 5W/m²K.
7.33 Forced convection heat transfer coefficient on base (left) and top (right) plates of a finless heat sink for a Ø59mm radial fan with $H_f = 4\text{mm}$ operating at a), b) 1000 rpm, c), d) 2000 rpm, and e), f) 4000 rpm under the influence of a crossing air flow of $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Contour level: $5\text{W/m}^2\text{K}$.

7.34 Normalised standard deviation (left) and minimum (right) fluctuations in $h_{fc}$ on the base plate of a finless heat sink for a radial fan with $H_f = 4\text{mm}$ and a), b) Ø15mm at 10,000 rpm; e), f) Ø59mm at 1000 rpm. Contour levels: a), c) 0.01; b), d) 0.03.

7.35 Forced convection heat transfer coefficient (left) and fluctuations in $h_{fc}$ (right) on the top surface (fan inlet side) of a finless heat sink for a Ø59mm radial fan with $H_f = 4\text{mm}$ operating at a), b) 1000 rpm, c), d) 2000 rpm and e), f) 4000 rpm under the influence of a crossing air flow of $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Contour levels: a), c), e) $2\text{W/m}^2\text{K}$; b), d), f) 0.01.

7.36 Forced convection heat transfer coefficient on the top surface (fan inlet side) of a finless heat sink for a Ø15mm radial fan with $H_f = 4\text{mm}$ operating at a) 5000 rpm and b) 10,000 rpm under the influence of a crossing air flow of $\overline{U}_{CF} = 0.21\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Contour level: $2\text{W/m}^2\text{K}$.

7.37 Cross over to an axisymmetric distribution in heat transfer coefficient for a) Ø15mm and b) Ø59mm radial fans.

7.38 Component locations and average heat transfer coefficient using a Ø15mm fan at 10,000 rpm. a) Component locations on the base plate for the case without cross flow or confinement. b) Corresponding component locations on the base plate for an example of $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. c) Average component heat transfer coefficient relative to the case without cross flow or confinement.

7.39 Mean radial distribution of Nusselt number on the base plate for a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm and under the influence of various cross flow velocities.
7.40 Area average Nusselt number on the base plate for a Ø15mm fan with $H_f = 4\text{mm}$ operating at a) 5000 rpm and b) 10,000 rpm under the influence of various cross flow velocities. ......................................................... 216

7.41 Area average Nusselt number on the base plate for a Ø59mm fan with $H_f = 4\text{mm}$ operating at 4000 rpm under the influence of various cross flow velocities. ......................................................... 217

7.42 Area average Nusselt number on the top plate for a) Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm and b) Ø59mm fan with $H_f = 4\text{mm}$ operating at 4000 rpm under the influence of various cross flow velocities. ............ 218

B.1 Calibration data for a number of temperature measurements ............ B-1
B.2 Orifice calibration charts ..................................................... B-2
B.3 Influence of spectral emissivity on emissive power. ..................... B-4
B.4 The general thermographic measurement configuration. ................. B-5
B.5 The relationship between emissive power and camera output signal. ... B-6
B.6 Calibration certificate for FLIR Indigo Merlin Mid infrared camera. .... B-8
B.7 Temperature distribution of a heated-thin-foil with a Ø15mm orifice. Contour level: 0.6K. .............................................................. B-9
B.8 Positioning stage, motor, and bearing housing .............................. B-10
## Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>a</td>
<td>Major heat sink channel dimension</td>
<td>m</td>
</tr>
<tr>
<td>a&lt;sub&gt;r&lt;/sub&gt;</td>
<td>Fan aspect ratio</td>
<td>-</td>
</tr>
<tr>
<td>A</td>
<td>Area</td>
<td>m&lt;sup&gt;2&lt;/sup&gt;</td>
</tr>
<tr>
<td>b</td>
<td>Minor heat sink channel dimension</td>
<td>m</td>
</tr>
<tr>
<td>B</td>
<td>IR calibration constant</td>
<td>-</td>
</tr>
<tr>
<td>Bi</td>
<td>Biot number</td>
<td>-</td>
</tr>
<tr>
<td>c</td>
<td>Chord length</td>
<td>m</td>
</tr>
<tr>
<td>C</td>
<td>Capacitance rate</td>
<td>W.K</td>
</tr>
<tr>
<td>C&lt;sub&gt;i&lt;/sub&gt;</td>
<td>1&lt;sup&gt;st&lt;/sup&gt; radiation constant</td>
<td>W.&lt;mu&gt;/m&lt;sup&gt;2&lt;/sup&gt;</td>
</tr>
<tr>
<td>C&lt;sub&gt;2&lt;/sub&gt;</td>
<td>2&lt;sup&gt;nd&lt;/sup&gt; radiation constant</td>
<td>µm.K</td>
</tr>
<tr>
<td>Cp</td>
<td>Specific heat capacity</td>
<td>J/kg.K</td>
</tr>
<tr>
<td>d&lt;sub&gt;b&lt;/sub&gt;</td>
<td>Hub diameter</td>
<td>m</td>
</tr>
<tr>
<td>D</td>
<td>Diameter</td>
<td>m</td>
</tr>
<tr>
<td>D&lt;sub&gt;h&lt;/sub&gt;</td>
<td>Hydraulic diameter</td>
<td>m</td>
</tr>
<tr>
<td>D&lt;sub&gt;i&lt;/sub&gt;</td>
<td>Difference between correct and measured temperature</td>
<td>K</td>
</tr>
<tr>
<td>E</td>
<td>Radiance emittance</td>
<td>W/m&lt;sup&gt;2&lt;/sup&gt;&lt;mu&gt;</td>
</tr>
<tr>
<td>E&lt;sub&gt;R&lt;/sub&gt;</td>
<td>Error relative to 1000 sample</td>
<td>%</td>
</tr>
<tr>
<td>E&lt;sup&gt;+&lt;/sup&gt;</td>
<td>Energy rate</td>
<td>J/s</td>
</tr>
<tr>
<td>f</td>
<td>Friction factor</td>
<td>-</td>
</tr>
<tr>
<td>f&lt;sup&gt;c&lt;/sup&gt;</td>
<td>Frequency</td>
<td>Hz</td>
</tr>
<tr>
<td>f&lt;sup&gt;+&lt;/sup&gt;</td>
<td>Cut-off frequency</td>
<td>Hz</td>
</tr>
<tr>
<td>f(Pr)</td>
<td>Function of Prandtl number</td>
<td>-</td>
</tr>
<tr>
<td>F</td>
<td>IR calibration constant</td>
<td>-</td>
</tr>
<tr>
<td>Gr</td>
<td>Grashof number</td>
<td>-</td>
</tr>
<tr>
<td>h</td>
<td>Heat transfer coefficient</td>
<td>W/m&lt;sup&gt;2&lt;/sup&gt;K</td>
</tr>
<tr>
<td>H</td>
<td>Height</td>
<td>m</td>
</tr>
<tr>
<td>H&lt;sub&gt;f&lt;/sub&gt;</td>
<td>Fan profile height</td>
<td>m</td>
</tr>
<tr>
<td>H&lt;sub&gt;CF&lt;/sub&gt;</td>
<td>Distance from confinement plate to fan inlet</td>
<td>m</td>
</tr>
<tr>
<td>I</td>
<td>Current</td>
<td>A</td>
</tr>
<tr>
<td>I&lt;sup&gt;+&lt;/sup&gt;</td>
<td>Turbulence intensity</td>
<td>-</td>
</tr>
<tr>
<td>k</td>
<td>Thermal conductivity</td>
<td>W/m.K</td>
</tr>
<tr>
<td>L&lt;sup&gt;*&lt;/sup&gt;</td>
<td>Length of channel</td>
<td>m</td>
</tr>
<tr>
<td>L&lt;sup&gt;+&lt;/sup&gt;</td>
<td>Thermal entrance coordinate</td>
<td>-</td>
</tr>
<tr>
<td>L&lt;sup&gt;'&lt;/sup&gt;</td>
<td>Hydrodynamic entrance coordinate</td>
<td>-</td>
</tr>
<tr>
<td>L&lt;sub&gt;c&lt;/sub&gt;</td>
<td>Characteristic dimension</td>
<td>m</td>
</tr>
<tr>
<td>m</td>
<td>Blending parameter</td>
<td>-</td>
</tr>
<tr>
<td>m&lt;sub&gt;+&lt;/sub&gt;</td>
<td>Mass flow rate</td>
<td>kg/s</td>
</tr>
<tr>
<td>M</td>
<td>Mach number</td>
<td>-</td>
</tr>
<tr>
<td>n</td>
<td>No. of channels</td>
<td>-</td>
</tr>
<tr>
<td>n&lt;sub&gt;p&lt;/sub&gt;</td>
<td>No. of IR recordings</td>
<td>-</td>
</tr>
<tr>
<td>n&lt;sub&gt;p&lt;/sub&gt;</td>
<td>No. of pixels in interrogation region</td>
<td>-</td>
</tr>
<tr>
<td>N</td>
<td>No. of samples</td>
<td>-</td>
</tr>
<tr>
<td>NTU</td>
<td>Number of transfer units</td>
<td>-</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
<td>-</td>
</tr>
<tr>
<td>P</td>
<td>Power</td>
<td>W</td>
</tr>
<tr>
<td>P&lt;sup&gt;c&lt;/sup&gt;</td>
<td>Pressure</td>
<td>Pa</td>
</tr>
<tr>
<td>P&lt;sub&gt;c&lt;/sub&gt;</td>
<td>Perimeter of channel</td>
<td>m</td>
</tr>
<tr>
<td>Pr</td>
<td>Prandtl number</td>
<td>-</td>
</tr>
<tr>
<td>q&lt;sub&gt;W&lt;/sub&gt;, Q</td>
<td>Heat transfer rate</td>
<td>W</td>
</tr>
<tr>
<td>q&lt;sup&gt;+&lt;/sup&gt;</td>
<td>Heat flux</td>
<td>W/m&lt;sup&gt;2&lt;/sup&gt;</td>
</tr>
</tbody>
</table>
\( \dot{q} \)  
Heat generation per unit volume \( \text{W/m}^3 \)

\( Q_{\text{input}} \)  
Power input \( \text{W} \)

\( \dot{Q} \)  
Volumetric flow rate \( \text{m}^3/\text{s} \)

\( R \)  
Thermal resistance \( \text{K/W} \)

\( R \)  
IR calibration constant

\( Ra \)  
Rayleigh number

\( Ri \)  
Richardson number

\( Re \)  
Reynolds number

\( s \)  
Fin spacing \( \text{m} \)

\( t \)  
Thickness \( \text{M} \)

\( t \)  
Time \( \text{s} \)

\( T \)  
Temperature \( \text{K} \)

\( T \)  
Torque \( \text{Nm} \)

\( \Delta T^* \)  
Non-dimensional fluctuating temperature amplitude

\( u_w \)  
Relative uncertainty

\( u,v,w \)  
Components of velocity \( \text{m/s} \)

\( u',v',w' \)  
Components of velocity fluctuations \( \text{m/s} \)

\( U \)  
IR camera output signal \( \text{A.U.} \)

\( U \)  
Velocity magnitude \( \text{m/s} \)

\( V \)  
Voltage \( \text{V} \)

\( V \)  
Velocity \( \text{m/s} \)

\( W \)  
Width \( \text{m} \)

\( W \)  
Result disregarding uncertainty in measured variables

\( x,y,z \)  
Cartesian coordinates \( \text{m} \)

\( X_o \)  
Unheated starting length \( \text{m} \)

\textbf{Greek}

\( \alpha \)  
Exit flow angle, swirl angle (axial fan), whirl angle (radial) \( (^\circ) \)

\( \beta \)  
Diffuser angle \( (^\circ) \)

\( \gamma \)  
Fan blade angle \( (^\circ) \)

\( \Delta \)  
Difference

\( \delta W \)  
Variance in the result

\( \varepsilon \)  
Emissivity

\( \varepsilon \)  
Channel aspect ratio

\( \varepsilon_{\text{eff}} \)  
Effectiveness

\( \lambda \)  
Wavelength \( \mu\text{m} \)

\( \mu \)  
Dynamic viscosity \( \text{kg/m.s} \)

\( \rho \)  
Density \( \text{kg/m}^3 \)

\( \sigma \)  
Standard deviation

\( \sigma_h \)  
Normalised fluctuation in \( h_{fc} \)

\( \sigma_{sb} \)  
Stefan-Boltzmann constant \( \text{W/m}^2\text{K}^4 \)

\( \tau \)  
Transmissivity

\( \tau_c \)  
Time constant \( \text{s} \)

\( \tau_w \)  
Wall shear stress \( \text{Pa} \)

\( \tau_{xy} \)  
Reynolds shear stress \( \text{kg/m.s}^2 \)

\( \phi \)  
Flow coefficient

\( \psi \)  
Pressure coefficient

\( \omega \)  
Rotational speed, rotational direction \( \text{rpm} \)

\( \omega \)  
Angular frequency \( \text{rad/s} \)

\textbf{Subscripts}

\( aw \)  
Adiabatic wall

\( abs \)  
Absolute

\( app \)  
Apparent
**Acronyms**

- **CCD**: Charge-coupled-device
- **FOV**: Field of view
- **FPA**: Focal plane array
- **IR**: Infrared
- **NUC**: Non-uniformity correction
- **PDF**: Probability density function
- **PIV**: Particle image velocimetry
- **RES**: Resolution
Chapter 1

Introduction

In this chapter, the motivations of this research thesis which surround the anticipated movement towards miniaturisation of cooling solutions are discussed. This is followed by a review of literature that is relevant to the area of thermal management and the advancements made in electronics cooling along with other disciplines. The objectives are outlined with the aim of addressing certain shortcomings relating to thermal management using active cooling methods particularly for portable device and low profile applications. The structure of this thesis is then summarised in the final section of this chapter.

1.1 Motivations

The technological advances in integrated circuits over the past number of decades has resulted in the evolution of electronic systems. This process of evolution constantly adapts to the technological and market requirements over time. The discussion of the advancements in integrated circuits is generally initiated by referencing the original forecast by Moore (1965) who anticipated that the number of transistors on an integrated circuit would double every year for the following ten years, or to 1975. This prediction in technological advances was considerably accurate over this time period. In 1975, Moore revised his prediction to a doubling of transistors on integrated circuits every two years, which has shown remarkable agreement even in most recent times. The consistant growth rate has resulted in the substantial advancements in functionality and affordability of computing
systems which are available in the commercial market. Similarly, the evolution of thermal
management in electronic systems has been inevitable due to the rising heat flux inherent
to integrated circuit designs. Increasing transistor density and switching speed of proces-
sors, combined with the dependency of failure mechanisms in integrated circuits on spatial
temperature gradients, temperature cycling, and rate of temperature change (Rogers and
Eveloy, 2004), has shifted thermal management from an afterthought in electronic system
designs, to the forefront in the device design process. This is particularly evident when
considering the thermal challenges in next-generation electronic systems as summarised
by Garimella et al. (2008). Diverse areas where critical issues exist and where research
is required were discussed in order to highlight the possible barriers on the progress of
 technological development. Device performance increases have therefore resulted in simult-
aneous advancements in thermal management solutions in order to maintain the rate of
 technological improvements in the industry.

In more recent times, the emergence of the portable electronic device market has also
resulted in the integration of cooling solutions which are required to dissipate adequate heat
with additional dimensional and power usage constraints imposed. With such limitations,
the design of thermal management solutions has become a science as opposed to an art
(Bejan and Lorente, 2008), where optimisation is an essential design criterion. The first
laptop to use active cooling to enhance heat dissipation was in 1995, where a thermal
design power of 6W was necessary (Miyahara, 2009). This featured an axial fan design
with an aluminium housing which also acted as a heat sink to conserve space. Since then,
various combinations of fan - heat sink designs have been incorporated into the mobile
personal computer to deal with increasing heat loads and maintain reliability. The diversity
of portable electronic devices, compared to other computer segments that generally have
standard configurations, consistently presents challenges for the thermal designer. Low
profile and weight device designs from one manufacturer can be completely different to
another (Mongia et al., 2008) and therefore continuous improvements are necessary to
determine the least imperfect (Bejan and Lorente, 2008) cooling solution for a given set of
constraints.

In addition to the thermal demands of increasing integrated circuit performance, the
motivations behind this research are also based on current and future consumer demands. Figure 1.1 presents the estimated global personal computer (PC) sales which includes PC servers, desktop PCs, laptops or notebooks, and netbooks or mini-notebooks from 1995 to 2014 (Juliussen and Feegan, 2010). The estimated contribution of mobile PCs (notebooks and netbooks) to the total sales is also shown, indicating that the overall growth in PC sales is driven by the demand for mobile devices. In 1995, mobile PCs were approximately 20% of the market share, whereas in 2014 this is expected to reach almost 70%. This is mainly attributed to both affordability and desktop-level performance of mobile PCs of recent. The increase in performance subsequently demands greater heat dissipation in lower form factor devices which are generally restricted to air cooling to maintain portability, unlike stationary computers which can integrate other cooling technologies such as liquid cooling loops to provide an order of magnitude increase in heat transfer coefficient (Wilson and Simons, 2005). However, although alternative cooling solutions are becoming available for large scale applications, air cooling remains the conventional choice due to benefits in cost along with a non-corrosive and abundant working fluid. Low profile cooling solution designs can also have a place in such applications as enclosures become densely packed with printed circuit boards and system air flow is less affected by thermal solutions which occupy less space.

Interestingly, mobile PCs are expected to account for approximately 46% of all portable

Figure 1.1: The personal computer industry: past, present and future trends, replotted from Juliussen and Feegan (2010).
electronic devices from 2010 to 2014, also highlighting the substantial growth of other portable devices such as smartphones (Kevorkian et al., 2010). Due to the technological advances at this miniature scale, it is projected that other single function devices, such as mobile internet devices, may eventually be phased out as portable handheld devices increase functionality to cover a wide range of applications on a single handset (Kevorkian et al., 2010). The increasing performance and multi-functional requirements of portable devices will therefore lead to greater heat flux densities within the device. This is also coupled with other design requirements such as maintaining low profile dimensions, extended battery life, and user comfort.

Currently, these handheld devices rely on passive cooling methods to maintain reliable operating temperatures. Typically this method provides a primary path of heat flow from the heat source to the circuit board and surrounding conductive layers until finally spreading to the outer case where natural convection and radiation dissipate heat from the device to an ambient environment (Langari and Hashemi, 2000). In situations where a body of lower temperature is in contact with the device case, heat is transferred through conduction. This path of thermal resistance can be quite complex due to the anisotropic materials within the portable device (Luo et al., 2008). Furthermore, designing a path of low thermal resistance to the outer casing of the device is limited when considering the maximum allowable surface temperatures of a handheld device to maintain user comfort. Berhe (2007) determined that this limit was 45°C for a plastic device casing. This results in case to ambient temperature differences of typically 15 - 20°C. Coupled with decreasing mobile device dimensions reducing convective surface area, and heat transfer coefficients from passive cooling, heat dissipation levels are becoming restricted.

Effective methods for extending the limits of this process by considering the implementation of phase change materials (Tan and Tso, 2004; Fok et al., 2010), micro heat pipes (Langari and Hashemi, 2000), and thermo electric coolers (Wilson and Simons, 2005; Garimella et al., 2008) are evident in the literature not only for portable applications, but also higher power examples (Fleischer et al., 2008; Garimella et al., 2008). Phase change materials absorb heat loads through latent heat of transformation which can then be released to the surroundings when the device is not in use. However, to achieve effective
cooling the device must only operate at increased thermal loads for the time required to complete phase transformation (Fok et al., 2010). Heat transfer performance would then revert to a reliance on the thermally resistant path to the outer casing if thermal loading was for extended periods of time. Micro heat pipe and thermo electric coolers can transfer heat away from the source but still require an effective sink to actively maintain acceptable component temperatures. Heat transfer densities using natural convection cooling are lower than that of forced convection over a given length scale. Consequently, methods of adequately coping with the increasing heat flux densities in portable electronics may be achieved using forced convection cooling. Transferring the majority of heat at the source to the ambient surroundings by forced convection also reduces the adverse effect on case temperatures. However, cost, space requirements, weight, and power usage are just some of the constraints associated with this method.

Miniaturisation of fan and heat sink assemblies are one possible solution to initiate forced convection within space constrained environments. This area of research is relatively new, despite the potential gains in thermal management. This is mainly due to the constraints associated with the implementation of the technology as previously outlined, combined with the absence of previous investigations into the heat transfer and fluid dynamic performances of miniaturised air cooling assemblies. An investigation and optimisation of such miniature forced convection cooling solutions could inevitably address some of the associated constraints in portable device applications and present miniature forced convection cooling solutions as either a suitable alternative or hybrid to the current cooling technologies. In addition, although forced convection cooling using larger scale fan assemblies is established in the literature, it is anticipated that research into the scaling of cooling solutions towards miniaturisation will also prove valuable in the understanding of fluid mechanisms and heat transfer for macro scale designs. In the following section, the literature related to forced convection cooling methods considered in this research thesis are discussed.
CHAPTER 1. INTRODUCTION

1.2 Literature

1.2.1 Thermal management

Heat sinks are widely used for maintaining reliable component temperatures in electronics cooling, whether it is by natural convection or forced convection. The main purpose of a heat sink is to increase the convective surface area of the heat source being cooled. This is typically achieved by introducing plate fins which have a large surface area to volume ratio to maximise convective surface area and minimise flow impedance. Consequently, this increases the level of heat dissipation once the overall heat transfer coefficient is not excessively deteriorated due to the increased resistance to fluid flow through the heat sink. This signifies that an optimum packing of fins into a heat sink volume exists which can maximise heat transfer. The optimum number of fins for various geometries that can be packed into a volume has been considered in the literature (Bejan and Lorente, 2008; Muzychka, 2005). Heat dissipation levels can also be augmented by increasing the heat transfer coefficient. In forced convection air cooling, this is generally achieved by increasing the mass flow rate through the heat sink, using fans which are compact and easily implemented. Another method of increasing both the convective surface area and heat transfer coefficient is by introducing finned designs which promote flow unsteadiness and turbulence within the heat sink. The introduction of pin-fin arrays is within this category as the fluid flow is no longer directed longitudinally as in plate-fin assemblies, but is allowed to interact in many directions (Li et al., 2005; Dogruoz et al., 2005; Peles et al., 2005). Thermal designers are looking further into modifications to finned designs which can be beneficial on heat transfer.

Plate-fin designs have also been adapted to produce unsteadiness within the heat sink channels. Some modifications to the conventional design include cross-cut (Kim and Kim, 2009), staggered (Leon et al., 2004), along with even more complex louvered and slit fin designs (Yang et al., 2007). The most obvious result of such designs is the interaction of air flow between the rectangular channels of the plate-fin arrays, in addition to the increased unsteadiness produced as air flows over the uneven plate surfaces. For cross-cut designs,
Kim and Kim (2009) assessed the effect of the number of cross-cuts on thermal performance for a parallel flow condition. The experimental results indicated that increasing the number of cross-cuts from one to three resulted in an increase in thermal resistance, even above that of the standard plate-fin design for the maximum number of cross-cuts considered. Based on the proposed correlations for cross-cut designs, and the correlations for plate-fin (Muzychka and Yovanovich, 2009; Teertstra et al., 1999) and pin-fin designs (Kim et al., 2004), the authors presented the transition of optimised design choice from plate-fin to cross-cut to pin-fin geometries.

Leon et al. (2004) applied a numerical approach to investigate the possible enhancements for staggered heat sink designs by removing sharp corners and introducing a rounded fin shape to the staggered array. The rounded design could dissipate the same heat load at 60% of the pumping power required by a standard plate-fin design. However this beneficial effect on heat transfer and pumping power was only assumed for Reynolds number greater than 800. Yang et al. (2007) examined the influence of fin spacing for plate-fin and interrupted plate-fin designs. It was noted that the louvered and slit fin designs required larger fin spacing than the conventional plate-fin design, therefore providing increased heat dissipation levels for reduced surface convection area. This is beneficial on material requirements, however the complexity of these designs would suggest that the plate-fin design would remain the more cost effective option. Pressure drop characteristics indicated that as fin spacing decreased the modified designs imposed a substantial increase in impedance of air flow through the heat sink over the conventional plate-fin design.

Numerical studies by Yang and Peng (2008, 2009) on non-uniform fin heights and fin widths for pin-fin heat sink designs, and Fabbri (1998, 1999) on asymmetric fin profiles for longitudinal finned heat sink designs, indicate the levels of complexity being investigated by thermal designers. Dimensionally scaled down versions of complex finned designs can be even more costly to implement due to the manufacturing process and tolerances required. In addition, complex designs which impose increased pressure drop may not be favourable at the miniature scale which already has limited flow rates due to scaled down fan geometries. For forced convection cooling solutions to be viable in the portable market, the preferable solution would be one which is cost effective, dissipates adequate heat
loads, and consumes a sufficiently low amount of power. Interestingly, it is not a complex solution which could possibly meet these stringent requirements, but an unconventional finless design proposed by Walsh et al. (2007) for low profile applications. This design was shown to provide a thermal resistance of $7.5^\circ{C}/W$ in a volume of $4.56 \text{mm}^3$ with a profile height of just 4mm including the fan. These authors also contributed to the area of low profile thermal management using forced convection by examining combined radial fan - heat sink arrangements with fins (Walsh and Grimes, 2007; Walsh et al., 2008), finned and finless heat sink designs with a commercially available miniature centrifugal blower (Egan et al., 2009), and also examining active cooling within a mobile phone handset (Grimes et al., 2010).

Loh et al. (2001) considered small scale axial and radial fan air flows over a plate-fin heat sink design for application to portable electronics. However, with a profile height of 20mm the design investigated would be too cumbersome for implementation in most modern portable devices which are typically less than 10mm in height. Although there is a general shortfall of literature addressing implementation of forced air cooling into portable electronic devices, the previously referenced experimental studies by Walsh and Grimes (2007), Walsh et al. (2008), and Egan et al. (2009) examined cooling solutions with profile heights under 10mm. The first of these studies by Walsh and Grimes (2007) highlighted the importance of combined fan and heat sink design, as higher flow rates did not always yield higher performance. Instead, optimum performance was achieved for particular fan blade and heat sink fin arrangements. A similar study by Walsh et al. (2008) on a radial finned heat sink centred around a radial blower in the absence of a volute, compliments the previous findings. A number of blade designs were analysed and the effect of aligned or impinging flow on fins evaluated. Results showed that optimum performance was attained when blade and fin angles were set so that impinging flow was eliminated, i.e. flow leaving fan blades was aligned with heat sink fins. Also investigated in this study was the effect of rotor diameter within a set cooling solution volume. It was found that heat sink performance remained largely unchanged whilst varying rotor diameter, and inversely varying heat sink surface area. As a result, the concept of an optimum coefficient of performance was introduced since smaller fans require considerably less pumping power.
This suggests that an optimal solution can be achieved which will adequately deal with the thermal loads in portable electronic devices and lead towards more energy efficient solutions.

Egan et al. (2009) analysed one of the smallest commercially available centrifugal fans having a footprint area of 256mm$^2$ and 5mm profile height in conjunction with custom made finned and finless heat sinks. The results showed that heat transfer enhancements using the simple finless design at the miniature scale could be achieved over that of the conventional finned design at lower fan speeds. Particle image velocimetry (PIV) was also used to obtain the velocity profile within the heat sink channels. The results showed the finless heat sink outperformed the dimensionally similar finned heat sink design for the majority of fan speeds examined. Added benefits of lower manufacturing cost, reduced fouling issues, and lower weight were highlighted making the finless design a preferable solution for the cost driven and energy conserving portable electronics market.

Direct cooling of components using miniature fans is another method which could maintain reliable component temperatures and conserve space in portable and space restricted environments. Heat dissipation levels from components and discrete heat sources has received some attention in the literature. Experimental and numerical studies by Weinstein et al. (2004) and Harvest et al. (2007) have shown the thermal effects of neighbouring components on a printed circuit board with different temperatures and spacings under natural convection. The limiting component separation distances were defined for the cases examined which avoided a reduction in heat dissipation. Fleischer et al. (2004) examined a similar arrangement to Weinstein et al. (2004), however, for a vertically orientated printed circuit board under the influence of forced convection. The circuit board was positioned in a wind tunnel and for various test section velocities from 0 - 2.5m/s. The significance on thermal performance by positioning components in a side-by-side, or an above-below orientation was highlighted for both standard and copper clad board designs over the range of air velocities. The authors also provide optimum placement criteria for different temperature components on circuit board designs. Although the cited studies present detailed information to address neighbouring effects and board conduction on component thermal
performance, there is limited knowledge on the influence of miniature fan flows on component positioning.

Previous studies suggest that cooling solutions should be designed as an integrated unit with fan and heat sink performance characterised in combination (Walsh and Grimes, 2007; Walsh et al., 2008; Egan et al., 2009). This has seldom been considered in the experimental and numerical design and optimisation of heat sinks in the literature. Generally, the thermal performance of heat sinks and discrete components are characterised with the assumption of an ideal or uniform velocity field and can be significantly different from the velocity field produced by rotating fan designs which will be discussed in the following section. As a result, if the extent of non-uniform heat transfer rates, produced by highly three-dimensional air patterns is unknown in the design stages, premature component failure may result.

1.2.2 Fan designs

Rotating fan designs are typically divided into the categories of axial, centrifugal/radial, and mixed which is a combination of both axial and radial designs. In electronics, the axial and centrifugal/radial designs are often implemented into cooling solutions to promote forced convection. There are many designs within each category that alter performance characteristics (Bleier, 1997) but in general axial fan designs deflect air in an axial direction, whereas centrifugal/radial fan designs deflect air radially outwards.

The velocity distribution of the air flow leaving large scale axial fan outlets has been well documented. Yen and Lin (2006) examined three different designs including shrouded, shroudless, and a shrouded winglet-blade design to confirm relative performance enhancements of the design choices mentioned. Each design also had varied geometric parameters such as blade angle, number of blades, fan speed and hub to tip ratio. Consequently, common trends in air flow distribution revealed that such air flow patterns are likely to exist for the majority of axial fan designs. Air flow which expanded downstream of the fan exit flow plane was observed, and related to the discontinuity of the shroud which confined the flow to the axial direction. Yoon and Lee (2004) measured the velocity distribution in a
similar region for a forward swept axial flow fan design. Using stereoscopic PIV, a three-
dimensional profile of the flow structures was created using axial, tangential, and radial
velocity components. In plane velocity vectors were presented with out of plane velocity
contours, revealing the highly three-dimensional flow patterns that are produced from an
axial fan in operation. The expanding flow pattern continued to diverge at a constant angle
after the fan outlet. The conical shape of the jet results from centrifugal forces within the
jet inducing a radial flow component; and based on the literature reviewed is synonymous
with axial fan designs.

Estevadeordal et al. (2000) also used the PIV technique for investigating the instantane-
ous and time-averaged velocity field around the fan blade pressure and suction surfaces.
This was achieved by synchronising the digital PIV system components to the blade posi-
tion. By throttling the inlet flow to the fan, various fan operating points could be examined
to determine the effects of system resistance on the local velocity field. Both flow visualisa-
tion and velocity field data highlighted unsteady aerodynamic excitations such as parallel
blade wakes, axial streaks, in addition to pressure and suction side flow separation. At
the trailing edge of the fan blade, increasing pressure rise across the fan promoted flow
unsteadiness. Increased pressure rise also changed the stagnation point on the blade from
the leading edge for a recommended operating condition, to the pressure side of the blade.
This in turn can contribute to the generation of small eddies on the suction side of the
blade, leading to the unsteadiness observed at the trailing edge (Bleier, 1997). Velarde-
Suarez et al. (2002) experimentally investigated the unsteadiness at various locations near
the inlet and outlet of a variable pitch axial flow fan with a 600mm tip and 380mm hub di-
ameter. This type of design offered an increase in the range of operation while maintaining
fan efficiency. High levels of unsteadiness were observed at the hub and housing regions,
which were promoted for off-design operating conditions.

Understanding the fluid flow patterns and locations of high fluid velocity and turbu-
lence intensities in the exit flow of an axial fan is also beneficial for the design of thermal
management solutions. Up to recently, local heat transfer distributions, resulting from fan
velocity patterns, have lacked documentation. An axial fan impinging air on a heated flat
plate can provide information on high heat transfer regions that correspond to the exit flow
distributions of the axial fan. This can be critically important for electronic systems where parallel flow is the common initial assumption. Sui et al. (2009a) investigated the exit flow patterns of an axial flow fan operating in standalone, and also in the presence of a flat plate positioned downstream and normal to the fan outlet. The existence of two shear layers was noted, using PIV in the downstream region. The outer shear layer resulted from the interaction of the fan exit flow with the surrounding ambient fluid. An inner shear layer was also observed due to fluid interaction with the zero flow region beneath the fan hub. In the presence of a flat plate, flow recirculation occurred beneath the hub which reduced the static pressure distribution at a location on the plate corresponding to the fan central axis. The lowest static pressure, and greatest flow recirculation, was noted at a height to diameter distance from the plate of \( H/D = 0.6 \). This closed recirculating flow is also apparent for annular jet impingement as noted by Ichimiya (2003). Interestingly, it was found that for an equivalent \( H/D = 0.65 \), the recirculating effects produced a maximum in heat transfer in the central region of the annulus. In a separate study on flat plate heat transfer performance, Sui et al. (2009b) examined the static pressure and heat transfer distributions at local radial points along the impingement plate from the central axis of the fan. This one-dimensional analysis was considered for two locations at a distance from the impingement plate, and also for two different designs of axial fan. Geometric dimensions of both fans were similar, however one design included additional hub fins to investigate if air flow structures beneath the hub could be promoted. The static pressure coefficient was at a maximum corresponding to the high velocity air flow which leaves the blade tip as noted by Yen and Lin (2006) and Yoon and Lee (2004). The resultant stagnation point on the plate surface was also the location of the maximum local heat transfer, often noted in studies of an impinging annular jet on a heated flat surface (Ichimiya, 2003; Chattopadhyay, 2004).

A shrouded axial fan has an exit flow area which resembles a circular annulus due to the zero flow condition at the fan hub. Consequently, some similarities in local heat transfer may exist with the case of an annular jet impinging on a heated flat plate, which has been analysed previously (Ichimiya, 2003; Chattopadhyay, 2004). It should be noted however, that for these studies the fluid leaving the nozzle of the annular jet and approaching the heated plate solely consists of an axial velocity component. As previously mentioned, the
fluid leaving the annular area of an axial fan is highly three-dimensional, and studies by Grimes and Davies (2004) and Grimes et al. (2001) show that the flow is unsteady and swirling, with a dependency on static pressure rise. This type of fluid motion has been represented in impinging jets by the introduction of swirl generators (Lee et al., 2002; Alekseenko et al., 2007; Huang and El-Genk, 1998), and also in Taylor-Couette flow with axial flows (Wereley and Lueptow, 1999; Hwang and Yang, 2004).

Lee et al. (2002) experimentally investigated a turbulent swirling round jet impinging on a flat surface for various swirl rates and plate distances from the nozzle exit. Heat transfer characteristics were acquired using a thermosensitive liquid crystal sheet, sprayed onto a gold-coated polyester substrate sheet which was electrically heated. At low nozzle to plate spacing, the inclusion of a swirl generator provided increased heat transfer over the case with no swirl generator. This can be accredited to the angle of the approach velocity of the fluid relative to the plate orientation, as regions of high shear are extended in the radial direction over that created by a predominantly axial fluid velocity with no swirl. However, at large nozzle to plate spacing, inducing swirl no longer proved beneficial in dissipating heat as the fluid velocity had decreased substantially before interaction with the plate surface. Alekseenko et al. (2007) examined the velocity field downstream of an annular swirling jet at a constant distance from the impingement plate, for various swirl rates. Increasing swirl rate resulted in a reduction of the recirculation zone beneath the jet centreline. The conventional jet with zero swirl rate also achieved a higher velocity magnitude near the impingement plate over all swirling jets. This effect on the surface heat transfer measurements by Huang and El-Genk (1998) was also noted.

Similar to the literature on axial fans, large scale radial fan designs have also received attention as the design is widely used in many engineering applications such as industrial dryers, air conditioning units, room circulation, and roof and wall exhausts. At the smaller scale, the use of radial fans for the purpose of augmenting heat transfer is particularly evident in electronics due to the possibility of relatively compact designs which can be easily accommodated. The extended use of radial fans for fluid movement has resulted in detailed research into the performance attributes of many designs.

Wu et al. (2008) investigated the velocity field at inlet, outlet, and tip leakage planes
for a centrifugal design with seven unequally spaced blades that were also staggered at different angles along the blade span from hub to shroud. The authors present this design as an effective way to improve aerodynamic performance and reduce noise. The inlet flow was found to be stable as the bellmouth directed the inlet flow into the impeller core. High levels of positive and negative vorticity exist on fan outlet measurement planes indicating counter rotational vortices which were generated by the backward curved airfoil blades in rotation. The majority of the mass flow tended towards the impeller hub, with increased velocity fluctuations at the shroud side, aided by the small vortices created by the leakage flow near the impeller shroud. The influence of a scroll housing on the non-dimensional fan performance was noted as being insignificant at a certain flow coefficient, however below this point the scroll housing offered an increase in total pressure and efficiency, with a decrease in the same observed at the higher flow coefficients.

Yen and Liu (2007) used a phase-locked PIV technique to determine the outlet flow field of a shrouded radial fan design which has dimensions suitable to laptop sized electronic applications. Two planes are considered in detail, and the exit flow from the shroud is shown to exit at an off-angle to the fan housing. This was similarly noted by Egan et al. (2009) in a study of the flow entering miniature heat sinks which were positioned adjacent to a shrouded fan outlet.

A systematic scheme for the design and generation of a similar dimensioned compact forward curved radial fan design for use in laptop computers was considered by Lin and Huang (2002). In this study, combined computational fluid dynamics and experimental tools were used to investigate blade shape, blade inlet angle, and housing outlet geometry for enhanced fan performance characteristics. The blade inlet angle was determined based on the best alignment with the inlet flow. The housing was redesigned to enhance flow rate and also reduce acoustic emissions. Blade surface pressure and velocity field distributions were investigated numerically, highlighting the similarities in flow patterns which existed for all blade angles considered. Overall, the main objective was to successfully combine numerical and experimental approaches to determine the optimum design at a reduced cost.

Wolfram and Carolus (2010) also combined numerical and experimental techniques to investigate the unsteady flow field generated by a radial fan impeller. The link between the
unsteady flow field of an isolated impeller and tone generation at blade passing frequency was investigated in detail. Typically blade passing frequency should only be dominant in voluted designs where the interaction of the outlet flow and the volute cut-off results in this tonal noise (Shepherd and Lafontaine, 1993). The main contribution to tonal noise for isolated impeller designs has previously been shown to occur when rotating stall is apparent (Carolus et al., 2000; Mongeau et al., 1993). The numerical study by Wolfram and Carolus (2010) suggests that a pre-swirl on the fan inlet flow exists which results in the formation of an inlet vortex. The interaction of the rotating blades with this quasi-steady vortex results in strong fluctuations which eventually act as tones occurring at blade passing frequency. However, the existence of this type of vortex at the inflow is more commonly associated with another centrifugal fan design known as a vortex blower, or cross flow fan (Bleier, 1997). Porter and Markland (1970) examined this design of fan at various operating points by throttling the inlet flow to the fan. Flow visualisation using polystyrene beads mixed in water presented the internal vortex which was at its greatest strength at the zero flow condition.

The acoustic emissions of miniature radial fans ranging in diameters of 15 - 32mm has been investigated by Walsh et al. (2009a) for numerous fan profiles down to just 0.5mm. The application of the work was focused on handheld electronic devices, and as a result measurement procedures were designed to reflect this. It was determined that the acoustic measurements at the fan outlet were dominant over that measured above the inlet. An increase in tone corresponding to the blade passing frequency was attributed to the possible inconsistancy with the manufacturing process which could have produced a slightly imbalanced design. A new scaling law was introduced which accounts for rotational speed, diameter, height and aspect ratios between 0.06 and 0.16. Guidelines were also outlined for the design of miniature radial fans to minimise acoustic levels. The authors discussed the influence of a perforated cover above the fan inlet which accentuated a tone increase at the blade passing frequency. This may be due to the blockaging effect reducing the fan flow rate and causing increased velocity fluctuations. The effect of perforated covers above miniature fan inlets on cooling solution performance was examined by Walsh et al. (2008) for combined radial fan - finned heat sink thermal solutions. Eight different perforated
cover designs were considered to investigate the effect on both fan flow rate and thermal performance versus a design without a cover. It was found that flow rate was reduced by as much as 60% for one design with a corresponding reduction in Nusselt number of 50% predicted based on an empirically determined scaling relationship.

It is therefore apparent that inlet distortions arising from practical situations can deteriorate both fan and heat transfer performance significantly. Ariga et al. (1983) assessed the effect of total pressure distortion on fan performance characteristics. This was achieved by investigating three cases which included hub distortion, tip distortion, and circumferential distortion. Honeycomb distortion generators were inserted in the various positions at the fan inlet to generate a non-uniform inlet velocity with a uniform static pressure distribution. It was determined that circumferential inlet distortion provided the greatest degradation on the fan outlet flow and fan efficiency, with radial distortion also appearing to have an influence at the larger flow rates examined.

Wright and Dire (1983) also examined distorted inflows on a similar large scale radial fan with outer diameter of 762mm. A non-uniform approach velocity was introduced above the fan inlet for the backward curved, airfoil blade fan design with volute. For all conditions considered, the distorted inflow to the fan reduced efficiency and pressure rise performance. The onset of rotating stall was also evident with severely distorted inflows.

Aside from the effects of practical operating conditions, fan designs also suffer from another degrading effect on performance when geometrically scaled. Grimes et al. (2005) initially noted the adverse geometric scaling effect on the performance of an axial fan design. A datum fan design with a 120mm diameter was geometrically scaled down to 1/3, which indicated a reduction in fan efficiency. Quin and Grimes (2008) examined the same designs including a 1/20 scale of the same axial fan design for a range of blade Reynolds numbers from 283 to 39,700 based on chord length and blade velocity at the mid-span. Below a Reynolds number of 1980, a viscous scaling effect was observed, where fan performance was adversely affected and could no longer be determined by the non-dimensional flow and pressure coefficients of the datum fan. Neustein (1964) also determined a Reynolds number effect on axial fan performance to occur below 2000.

The miniaturisation of radial fan designs also results in similar scaling effects on fan
performance as shown by Walsh et al. (2009b) and Walsh et al. (2010). In the first study by Walsh et al. (2009b), the influence of fan profile scaling for fan diameters of 15 - 30mm was examined to address the issues associated with implementing miniature fan designs in low profile applications. The fan characteristics of flow rate, pressure rise, and power consumption were experimentally measured while varying the blade profile alone. A low Reynolds number effect was noted at 650 based on chord length and blade tip velocity which resulted in a reduction of flow rate, and a simultaneous increase in power consumption over that predicted using conventional scaling laws. At the miniature scale, these fan scaling laws were found to be valid only for fan aspect ratios between 0.12 and 0.17. In a separate study, Walsh et al. (2010) examined the same fan characteristics and range of fan diameters but in this case varying blade chord length. Similar trends in reduced fan performance were noted at low Reynolds numbers, and the authors applied simple boundary layer theory to determine the main contribution to this scaling effect for miniature fans. In doing so, the authors proposed an alternative empirical based correlation for determining the performance of radial fan designs operating at low Reynolds numbers.

A piezoelectrically actuated vibrating cantilever is another fluid mover which is often classified as a fan. Kimber et al. (2007) and Kimber and Garimella (2009) provided local heat transfer coefficient maps on a flat surface influenced by piezoelectrically actuated vibrating cantilevers, or piezoelectric fans. These fans also require limited space to operate and are therefore another possible cooling solution for low profile and portable electronics. In the first study by Kimber et al. (2007), variation of amplitude and gap distance between the fan and heat source was considered. Increasing gap distance resulted in lower maximum heat transfer coefficients; however a larger plate area was cooled. In addition, the impact of vibration, amplitude, and design parameters of piezoelectric fans on heat transfer over a flat surface was considered by Kimber and Garimella (2009). The effect of fan width in particular was assessed and shown to provide large gradients in heat transfer coefficient as fan width decreased. The results were expressed in terms of the ratio of gap distance to vibration amplitude, which governed the shape of the forced convection pattern on the flat surface. The above studies determined the complex trends in heat transfer non-intrusively using infrared thermography and a heated-thin-foil technique. Acikalin et al.
(2004) presented flow visualisation and heat transfer measurements of piezoelectric fans to determine the feasibility of these cooling solutions in cellular phone and laptop sized devices. A number of arrangements and fan orientations were considered including piezoelectric fans operating in tandem. It was concluded that piezoelectric fans could be viable for dimensionally constrained portable devices and also for larger scale devices such as laptops. However, these fans would only be able to supplement rotary fan designs in larger dimensioned portable devices.

In section 1.2.1, previous studies which considered complex finned designs to promote fluid unsteadiness within the channels of a heat sink were discussed. Much of the literature investigating these novel heat sink designs consider a uniform approach velocity. However, an unsteady outlet flow from rotating blades is evident when examining the findings of previous literature on fan outlet flows in this section. Unsteadiness due to the abrupt discontinuity of a fan blade at the blade tip can inhibit fan performance characteristics (Yen and Lin, 2006). In miniature heat transfer applications however, heat sink designs that conserve such fluid structures may be advantageous for increasing heat dissipation levels when considering the unconventional finless design first presented by Walsh et al. (2007). Therefore the introduction of complex finned designs to promote flow unsteadiness may be unnecessary for low profile cooling solutions. In figure 1.2, an example of the fluid structures generated by rotating a miniature radial fan of 15mm diameter and 4mm blade height is presented. The forward curved design was rotated above a flat surface for a Reynolds number of 115, based on the blade tip velocity and chord length as characteristic length scale. This experiment was conducted in a free environment without a volute, allowing air to exit radially in all directions. Flow visualisation and PIV were used to investigate the fluid structures at the exit of the miniature radial fan.

The flow visualisation image in figure 1.2 a) shows vortex pairs, generated by the rotating miniature fan blades, travelling in the radial direction. A time-average of 1000 velocity vector maps has been subtracted from an instantaneous vector field for figure 1.2 a) to produce a vorticity profile of the near exit region, shown in figure 1.2 b). This region is defined in figure 1.2 a) by X and Y. Figure 1.2 b) highlights the counter rotational nature of these vortices upon exiting the fan blades. The clockwise vortices in figure 1.2 a) are stretched by
CHAPTER 1. INTRODUCTION

Figure 1.2: Fluid structures a) and vorticity b) generated by a miniature radial fan operating at \( Re = 115 \). The experimental configuration is detailed in c).

the mean radial flow and consequently diffuse into the boundary layer over the flat surface. The vorticity profile indicates both the clockwise and counterclockwise vortices diffuse rapidly over the 15mm interval shown in figure 1.2 b). This flow visualisation and velocity field indicates the unsteady fluid flow which could be used to promote heat transfer in a finless design.

Miniature radial fan designs have been shown to experience a scale effect due to viscous dominance at low Reynolds numbers (Walsh et al., 2009b, 2010). A similar scaling effect has also been noted for axial fan designs (Neustein, 1964; Quin and Grimes, 2008). As this effect results in much lower flow rates than anticipated through conventional scaling laws, the creation of vortex structures and unsteadiness may be beneficial to promote fluid mixing and hence enhance heat transfer to a sufficient level at these scales.

In summary, a wide range of fan designs, both axial and radial, have been investigated through velocity field and fan performance analyses. The majority of the referenced studies are based on larger scale applications with fan diameters ranging 50 - 800mm. However, some work on miniature designs in terms of fan performance characteristics and acoustic emissions is evident due to the anticipated move towards miniaturisation previously discussed in section 1.1. Although the influence of miniaturisation on fan performance has now been documented, only preliminary studies exist on the thermal performance of combined miniature fan - heat sink cooling solutions. The finless heat sink concept has been
shown to provide thermal performance at a similar level to finned designs of equal exterior dimensions at miniature scales, however further investigation is required to understand the reason this unconventional design produces such enhancements. Similarly, it is also necessary to determine optimisation criteria for heat sink designs at this scale which has not yet been considered for combined fan - heat sink solutions. Even at the larger scales, there is an absence of work into the influence of unsteady and non-uniform velocity fields from fan assemblies on local thermal performance despite its significance in electronic cooling systems.

1.3 Objectives

This thesis examines the heat transfer performance of miniature and low profile cooling solutions which integrate rotating fan assemblies to promote forced convection cooling in space constrained environments. The primary objectives were to:

1. Examine the thermal performance, and hence suitability, of finless heat sink designs over conventional finned designs at miniature and low profile scales. In doing so, determine if alignment of the fan exit flow with the heat sink channels has a positive contribution to the overall heat transfer performance of integrated fan - heat sink cooling solutions.

2. Investigate the influence of heat sink profile scaling on the thermal performance of finned and finless geometries. In addition, implement a theoretical approach to complement the experimental investigation of profile scaling and to determine the design criteria for accurately predicting optimum thermal performance of low profile forced convection cooling solutions.

3. Develop experimental facilities and correction techniques for accurate local heat flux measurements.

4. Determine the significance of electronic component positioning on component thermal performance due to the unsteady and non-uniform flow field associated with both axial and radial fan designs.
5. Assess the fluidic mechanisms in finless heat sink designs which influence overall thermal performance and that correspond to variations in local surface heat transfer.

6. Investigate the practical operating conditions of fan inlet flow confinement and distortion by introducing crossing air flows at various confinement heights, and determine the implications on the finless heat sink flow field and thermal performance.

The findings from the above objectives can guide thermal designers to the possible gains in thermal performance which can be achieved in current and future cooling solutions of space restricted devices using forced convection methods.

1.4 Thesis structure

In the following chapter, a theoretical approach to determine heat sink performance is outlined which uses a suitable model from the literature to predict hydrodynamic and thermal performance within channels of arbitrary shape, combined with conventional heat exchanger theory applied to single fluid heat sinks. Chapter 3 and 4 detail the experimental methodologies to investigate global and local heat transfer and fluid flow characteristics. The results and discussion are contained within three chapters. Firstly, comparative analyses on finned and finless heat sink designs are presented in Chapter 5 using bulk heat transfer measurements. The influence of component positioning on overall component heat transfer performance is also examined for a radial fan design and finless heat sink geometry in this chapter. Chapter 6 examines the local heat transfer distribution due to axial fans impinging air normal to a flat surface in the absence of a heat sink. This was considered to assess direct component cooling in the absence of a heat sink which allows for profile heights suitable to low profile applications. In Chapter 7 the fluidic mechanisms due to radial fan designs within the finless channel, representative of a finless heat sink, are presented in addition to local heat transfer distributions. The second half of Chapter 7 is devoted to examining the influence of practical operating conditions that can be experienced when a cooling solution is introduced into an electronic device, and often overlooked in the design process. Finally, the conclusions and recommendations are reported in Chapter 8.
Chapter 2

Heat exchanger theory

This chapter presents the theory of heat exchanger design for forced convection single fluid applications. A general model is implemented to predict fluid dynamics and heat transfer performance for either finned or finless geometric designs. Conventional heat exchanger analysis is also employed to extend the prediction tool to heat sink applications in electronics cooling, where performance is commonly characterised by a driving difference in temperature between the heat sink and the inlet fluid.

2.1 Fluid flow configuration

The flow configuration within the majority of heat sinks reflects simultaneously developing hydrodynamic and thermal boundary layers and is particularly evident in this thesis where a radial fan or blower is positioned adjacent to the heat sink channels for various profile height designs. Optimisation relies on this consideration, as high heat transfer rates are achieved in the developing flow region compared to the developed region. A similar fan - heat sink arrangement, to that considered in the experimental analyses, is presented in figure 2.1 for a finned and finless geometry. The impeller rotational direction is noted, along with the resultant fluid momentum within the heat sink channels. A cross-sectional view of these channels, indicated in figure 2.1 as section A-A, is presented in figure 2.2, and describes the hydrodynamic and thermal boundary layer profiles within each heat sink geometry. The heat sink has an isothermal temperature, $T_w$, where $T_w > T_i$. The fluid,
Figure 2.1: A typical radial fan and heat sink arrangement for a) finned and b) finless designs.

Section A-A

Figure 2.2: Flow configuration along heat sink channels in both finned and finless designs

air, enters the fan inlet at temperature, $T_i$, and exits the heat sink channels at an elevated temperature, $T_o$.

Analytical and experimental solutions for such flow conditions in compact heat exchangers have been considered in the literature (Kays and London, 1984). This thesis utilises a model developed by Muzychka and Yovanovich (2004) for predicting heat transfer coefficients in non-circular ducts with simultaneously developing hydrodynamic and
thermal boundary layers. The individual channels of the finned and the finless heat sink geometries are representative of rectangular ducts. For a rectangular duct and an isothermal boundary condition, the dimensionless heat transfer is defined as (Muzychka and Yovanovich, 2004):

$$\text{Nu} = \left[ 2f(Pr) \right]^m \left[ \left( \frac{fRe}{L^*_{\sqrt{A}}} \right)^{\frac{1}{2}} + \left\{ \frac{0.6135}{8 \sqrt{\pi \epsilon}} \right\} \sqrt{\frac{L^*_{\sqrt{A}}}{A}} \right]^{m/5} \left[ \left( \frac{f \sqrt{Re \frac{1}{Pr}}} \right)^{\frac{1}{2}} + \left\{ \frac{3.24}{8 \sqrt{\pi \epsilon}} \right\} \sqrt{\frac{L^*_{\sqrt{A}}}{A}} \right]^{m/5} \left[ \frac{1}{m} \right]^{1/m}$$

(2.1)

where $f(Pr)$ is:

$$f(Pr) = \frac{0.564}{\left[ 1 + (1.664 Pr^{\frac{1}{3}})^{\frac{5}{3}} \right]}$$

(2.2)

The blending parameter, dependent on fluid Prandtl number is:

$$m = 2.27 + 1.65 Pr^{\frac{1}{3}}$$

(2.3)

and $\epsilon$, the duct aspect ratio, is $\frac{b}{a}$.

The fully developed friction factor Reynolds number, based on the square-root of the cross-sectional area as characteristic dimension $\sqrt{A}$, was found to be independent of fluid velocity and simply a function of aspect ratio.

$$fRe_{\sqrt{A}} = \frac{12}{\sqrt{\epsilon} (1 + \epsilon) \left[ 1 - \frac{192 \epsilon \tanh (\frac{\pi}{2})}{\pi^2} \right]}$$

(2.4)

The term $L^*_{\sqrt{A}}$ refers to the dimensionless thermally developing co-ordinate at the heat sink exit with the characteristic dimension, $\sqrt{A}$

$$L^*_{\sqrt{A}} = L/\sqrt{A} \frac{\nu L}{Re_{\sqrt{A}} Pr} = \frac{\mu L}{\left( \frac{m}{n} \right) Pr}$$

(2.5)

where $\dot{m}$ is mass flow rate, and $n$ denotes the number of channels in the heat sink. For the finless cases, $n$ equals 1. The mean heat transfer coefficient can be resolved using its
relationship with Nusselt number.

\[ h = \frac{Nuk}{\sqrt{A}} \]  

(2.6)

The system characteristics can be predicted through the relationship between pressure drop across the channel and shear stress or frictional effects.

\[ A\Delta P = \tau_w P_c L \]  

(2.7)

This is the momentum theorem in the longitudinal direction of the channel flow. The wall shear stress can also be represented through the non-dimensional friction factor.

\[ f = \frac{\tau_w}{\frac{1}{2} \rho U^2} \]  

(2.8)

Combining Eq. (2.7) and (2.8), the pressure drop across the heat sink can be represented as

\[ \Delta P = f \frac{P_c L}{A} \left( \frac{\rho U^2}{2} \right) \]  

(2.9)

Considering the entrance region effect on the dimensionless friction, an apparent friction factor is used in Eq. (2.9). This is defined below in Eq. (2.10) in the apparent friction factor Reynolds number (Muzychka and Yovanovich, 2004).

\[ f_{app} Re_{\sqrt{A}} = \left[ \left( \frac{3.44}{\sqrt{L^+/Re_{\sqrt{A}}}} \right)^2 + \left( f Re_{\sqrt{A}} \right)^2 \right]^{1/2} \]  

(2.10)

The dimensionless hydrodynamically developing co-ordinate at the heat sink exit is:

\[ L^+_{\sqrt{A}} = \frac{L}{\sqrt{A}} \frac{\mu L}{\left( \frac{m}{\rho} \right)} = \frac{\mu L}{\left( \frac{m}{\rho} \right)} \]  

(2.11)

As the non-dimensional hydrodynamic length is increased, the solution for the apparent friction factor Reynolds number approaches that of the fully developed friction factor Reynolds number, and entrance region frictional effects become insignificant versus the fully developed. Therefore the apparent friction factor can be used over the fully developed.
friction factor, $f$, as it represents the frictional effects for both flow conditions, depending on the hydrodynamic entrance position. Using Eq. (2.10), and representing the mean velocity in terms of mass flow rate, the relationship for pressure drop and flow rate across the heat sink is

$$
\Delta P = \frac{\dot{m}}{n} \left[ 1 - \frac{f_{\text{app}} R e_{\text{avg}} P_{c} L \mu_{c}}{\rho A^{3/2}} \right] 
$$

(2.12)

Eq. (2.12) is valid for both finned and finless geometries.

### 2.2 Heat exchanger model

Heat exchanger design primarily addresses both the heat transfer and fluid pumping power required to overcome pressure drop within the exchanger for multiple fluid applications. The heat transfer analysis can be approached using either the Log Mean Temperature Difference (LMTD) or Effectiveness – Number of Transfer Units ($\varepsilon$-NTU) methods. The LMTD method requires knowledge of both inlet and outlet fluid temperatures of the heat exchanger. The equivalent $\varepsilon$-NTU method has been considered appropriate to use here as it simplifies the analysis even for complex arrangements.

In figure 2.3 a) a two-fluid heat exchanger arrangement is presented. Conventional analysis is based around a two-fluid heat exchanger where the actual heat transferred between a hot and cold fluid is

$$
Q = [\dot{m}C_p]_h (T_{i,h} - T_{o,h}) = [\dot{m}C_p]_c (T_{o,c} - T_{i,c})
$$

(2.13)

The definition of heat exchanger effectiveness is the ratio of the actual heat transfer to the maximum possible heat transfer

$$
\varepsilon_{\text{eff}} = \frac{Q}{Q_{\text{max}}}
$$

(2.14)

In order to achieve the maximum heat transfer rate, the maximum possible temperature difference within the heat exchanger is required. The capacitance rate, $C = \dot{m}C_p$, is also a driving force for the maximum heat transfer rate. When calculating the maximum heat
transfers, there are two separate capacitance rates for the hot and cold fluids to consider. An energy balance between both fluids shows that the only fluid which can achieve the maximum temperature difference is the fluid with the minimum capacitance rate. Eq. (2.14) can be presented in terms of capacitance rate and temperature

\[ \varepsilon_{\text{eff}} = \frac{C_c(T_{o,c} - T_{i,c})}{C_{\text{min}}\Delta T_{\text{max}}} = \frac{C_h(T_{i,h} - T_{o,h})}{C_{\text{min}}\Delta T_{\text{max}}} \]  (2.15)

and the actual heat transfer can be expressed as a function of the heat exchanger effectiveness

\[ Q = C_{\text{min}}\Delta T_{\text{max}}\varepsilon_{\text{eff}} \]  (2.16)

The boundary conditions for a single fluid heat sink can be combined with the above two-fluid heat exchanger analysis to determine the heat transfer (Webb, 2007). It is assumed that the air passing through the heat sink represents the cold fluid, whereas the surrounding heat sink material represents the hot fluid, having a constant temperature, \( T_w \), from inlet to outlet, as shown in figure 2.3 b). The air therefore must have a capacitance rate relative to the outside material such that it cannot influence the isothermal condition. It is possible to consider an energy balance similar to Eq. (2.16) as the forced convection heat transfer is of interest. For energy to be conserved, as the heat sink approaches an isothermal condition

---

**Figure 2.3:** Layout of a a) multiple fluid heat exchanger and a b) single fluid isothermal wall heat exchanger.
CHAPTER 2. HEAT EXCHANGER THEORY

\[ T_{i,h} - T_{o,h} = (T_w - T_w) = 0, \]  
the capacitance rate must approach infinity. Hence, the fluid flowing through the heat sink will always remain the minimum fluid.

The number of transfer units (NTU) is defined as

\[ NTU = \frac{h A_{\text{conv}}}{\dot{m} C_p} \]  
(2.17)

where \( h \) is the forced convective heat transfer coefficient (Eq. 2.6), and \( A_{\text{conv}} \) is the surface area of the heat exchanger exposed to forced convection. The relationship \( C_{\text{min}}/C_{\text{max}} = 0 \) found here, results in the heat exchanger effectiveness relation approaching a simple equation (Holman, 2002).

\[ \varepsilon_{\text{eff}} = 1 - e^{(-NTU)} \]  
(2.18)

Finally using these parameters in Eq. (2.16), the heat transfer for a forced convection heat sink is

\[ Q = \dot{m} C_p (T_w - T_i) \left[ 1 - e^{\left( \frac{-h A_{\text{conv}}}{\dot{m} C_p} \right)} \right] \]  
(2.19)

In electronics cooling, the thermal performance of an isothermal forced convection cooling solution is generally expressed using thermal resistance which is based on a temperature difference between the heat sink and the fan inlet air temperature. Similarly, the heat transfer coefficient for the prediction model can be represented using the inlet temperature difference:

\[ h_{\text{ITD}} = \frac{Q}{A_{\text{conv}} (T_w - T_i)} \]  
(2.20)

Also \( h_{\text{ITD}} \) can be used to determine the Nusselt number based on inlet temperature difference.

\[ Nu_{\text{ITD}} = \frac{h_{\text{ITD}} D_h}{k} \]  
(2.21)

where \( D_h \) is the hydraulic diameter \( (4A/P_c) \) of the heat sink channel, and \( k \) is the thermal conductivity of air. The suitability of using the hydraulic diameter as the length scale in the
above equation over $\sqrt{A}$ is discussed in the proceeding results chapters. The development of Eq. (2.1), which uses $\sqrt{A}$ as an arbitrary length scale, captures the non-dimensional heat transfer for many non-circular geometries under one expression. In the current study, only one elemental geometry is considered, however the theory presented above can be applied to many elemental and complex geometries.

2.3 Closure

An analysis of the combined entrance and fully developed regions of non-circular ducts was implemented to predict the performance of miniature heat exchangers. Eq. (2.12) was proposed to determine the pressure drop from heat exchanger inlet to outlet and independent of heat exchanger geometry. The conventional heat exchanger model for two-fluid designs was applied to a single fluid design with an isothermal boundary representing a heat sink for electronics cooling. This method provides a relationship in Eq. (2.19) for determining the forced convective heat transfer rate of a heat sink and can be compared to the experimental investigations that will be discussed in the following chapter. Based on this comparison, the validity of the prediction method presented in this chapter will be assessed for suitability in designing combined fan - heat sink arrangements including finned and finless geometries.
Chapter 3

Experimental techniques and facilities

This chapter describes the experimental methods and facilities used to investigate fluid dynamics and heat transfer for miniature and low profile forced convection cooling solutions. Comprehensive data acquisition and analysis procedures were developed for the miniature scales examined, thus increasing confidence levels in measurement accuracy. Facilities for acquiring bulk fluid flow and heat transfer information were developed to aid in the understanding of performance enhancements which can be achieved at this scale.

3.1 Mean heat transfer measurements

Bulk isothermal heat transfer measurements were conducted on miniature and low profile cooling solution designs which utilised commercially available radial flow fans. Through investigation of various cooling solution designs with a commercially available radial fan, it was possible to highlight heat transfer enhancements which could be achieved through relatively simple design choices favouring portable applications. The first section presents the experimental arrangement for determining the heat transfer performance of both finned and finless cooling solutions, where the fan outlet flow is aligned with the heat sink channels through the introduction of a diffuser. The second section details the experimental method used to examine the influence of reducing profile height, for both finned and finless geometries, on heat transfer performance. Finally, mean heat transfer measurements were also carried out on an isothermal component, designed to immitate a real electronic component.
This was positioned at various distances from a radial flow fan. The apparatus employed in positioning these discrete, isothermal components relative to a radial fan is also included.

### 3.1.1 Miniature cooling solutions

The experimental configuration consisted of a Micronel fan, a flow diffuser and the heat sink which was designed using optimum parallel plate spacing theory presented by Bejan and Sciubba (1992). The fan tested for the current work is the Micronel U16LM-9 having a footprint area of 256mm$^2$ and a profile of 5mm. This is currently one of the smallest commercially available fans for forced convection cooling of electronics. The pressure versus flow rate characteristics are supplied by the manufacturer for a nominal speed of 6000 rpm. In designing the finned heat sink the pressure versus flow rate characteristics at a fan speed of 8000 rpm were used. The manufacturer’s data were scaled using the fan scaling laws detailed in Bleier (1997) and defined here in Eq. (3.1) and (3.2).

\[ \dot{Q} \propto \omega \]  
\[ \Delta P \propto \omega^2 \]  

Figure 3.1 shows a schematic of the heat sink highlighting the variables that were optimised; these include the fin spacing ($s$) and fin thickness ($t_{fin}$). In determining the optimum fin spacing, the pressure drop ($\Delta P$) across the bank of channels, was set equal to 6.5 Pa so that the fin spacing of the heat sink was optimised for a fan speed of 8000 rpm and should correspond to a total volumetric flow rate of approximately $2.5 \times 10^{-4} \text{m}^3/\text{s}$ passing through the heat exchanger. In order to achieve this, a fin spacing of approximately 1.1mm was required. The second parameter examined was the thickness of the fins, $t_{fin}$. This was optimised by ensuring that the fin efficiency was greater than 99% and Ellison (1989) shows this to be the case when Eq. (3.3) is satisfied.

\[ t_{fin} \geq \frac{40hH}{k_{HS}} \]
Figure 3.1: Schematic of a) finned and b) finless heat sinks outlining the overall dimensions and the optimised parameters $s$ and $t_{fin}$.

The flow condition within the heat sink reflects an underdeveloped duct flow entrance region, which becomes fully developed at the heat sink exit. A value of $60\text{W/m}^2\text{K}$ was chosen for $h$ in Eq. (3.3) based on an empirical Nusselt number relationship for this flow condition (Holman, 2002). Using copper as the heat sink material, Eq. (3.3) reveals that the fin thickness should be at least 0.06mm. However, the technique used to manufacture the heat sink required this dimension to be 0.3mm, hence ensuring a more than adequate fin thickness. This also provided for a large tolerance if any discrepancies in the choice of $h$ for Eq. (3.3) were apparent. Any discrepancies in the choice of $h$ were found to have no influence on the choice of fin thickness which was set by the manufacturing limitations.

The result was that a finned heat sink with a maximum of six channels could be manufactured while maintaining the heat sink footprint constraints that were applied. The finless heat sink was manufactured to have the identical specifications of profile (4mm), footprint area 200mm$^2$, and external wall thickness (1mm) and is shown in figure 3.1.

As previously mentioned in the literature review, in order to achieve the optimum thermal performance of a fan and finned heat sink in parallel, flow must enter the heat sink aligned with the heat sink walls and fins as noted by Walsh et al. (2008). Figure 3.2 presents the backward curved Micronel fan design with its top casing removed to show the blade and volute design. Also included is a schematic of a velocity vector diagram for this design.
with a rotational speed, $\omega$, and radial velocity, $V_R$. The angle of the flow exiting the fan casing can be measured using particle image velocimetry (PIV) and is dependent on system resistance. A high system resistance such as that created by a finned heat sink will result in a lower flow rate through the fan. This in turn will result in reducing only the radial flow component and hence the angle at which fluid leaves the fan. It can be seen that high system resistances will reduce $\alpha$, whilst low system resistances will result in increasing this angle. In order to align the flow exiting the fan with the heat sink channels, a number of diffusers with angles varying between $25^\circ$ and $60^\circ$ with both curved and straight walls were manufactured from polycarbonate. As discussed above, it was found that the optimum angle for flow alignment varied between the finned and finless case. Each diffuser was tested using PIV to measure the velocity of the flow in each of the heat sinks and it was determined from the measurements that a curved diffuser with $\beta = 25^\circ$ and a straight diffuser with $\beta = 50^\circ$ resulted in flow alignment within the finless and finned heat sinks respectively. The experimental arrangement using PIV to examine the optimum angle of flow alignment is included in the following chapter on advanced experimental techniques.

During experimentation the top cover of the heat sink, as shown in figure 3.3, was in position so as to create a fully closed channel. This cooling solution has a footprint of $256\text{mm}^2$ for the fan, either $231\text{mm}^2$ for the finless or $278\text{mm}^2$ for the finned heat sink and
diffuser combination, and a profile of only 5mm. The heat source used for experimentation was a 6×6mm Minco thin-film heater, and was attached to the base of the heat sinks. The fan and thin film heater were powered using a TTi dual DC power supply and rotational speeds ranging from 1000 rpm to 8500 rpm were measured using an Omega HHT13 optical tachometer. Three 0.25mm K-type thermocouples, calibrated to 0.1K, were attached using a heat resistant tape to the centre of the top, base and left wall of the heat sink. The ambient air temperature at the fan inlet was also recorded. All thermocouples were connected to a National Instruments 9211 USB High Speed Carrier and temperatures were recorded and plotted using LabVIEW 8. During experimentation, the entire heat sink was found to be almost isothermal, as a maximum temperature difference between the hottest (base) and coldest (top) surface of approximately 5% was measured. As the experiment was conducted under an almost isothermal condition, any temperature deviations in the lateral direction were considered negligible due to the worst case comparison of the heated base with the unheated top surface only producing this 5% difference in measured temperature.

Also of significance, was that the external surfaces of the heat sink were covered using insulating tape to reduce the heat loss due to secondary cooling mechanisms such as natural convection and radiation. The purpose of this was to attempt to simulate the operational environment for such a system in a typical portable electronic device where the total set-up is likely to be confined within a very small cavity.

Figure 3.3: Finned (top) and finless (bottom) cooling solutions employing straightening diffusers at the fan exit.
In electronics cooling, the most widely used thermal performance indicator is the thermal resistance and is based on inlet temperature difference, as defined in Eq. (3.4)

\[
R_{\text{tot}} = \frac{(T_w - T_i)}{Q_{\text{input}}} \quad (3.4)
\]

In the above expression, \(Q_{\text{input}}\) is the power supplied to the thin film heater, \(T_w\) is the steady state wall temperature, and \(T_i\) is the ambient inlet air temperature. The wall temperature was calculated by averaging the temperature readings recorded over the entire heat sink. Eq. (3.4) presents the total thermal resistance of the heat sink. Heat dissipated through forced convection is of interest and is independent of the application. Therefore it is necessary to account for the other mechanisms of heat transfer, such as natural convection, conduction, and radiation, which shall be referred to as secondary heat transfer modes. The contribution of these secondary heat transfer modes to the overall heat dissipated is represented by the term \(Q_{\text{Losses}}\) below.

\[
Q_{\text{input}} = Q_{fc} + Q_{\text{Losses}} \quad (3.5)
\]

Hence, the forced convection thermal resistance is

\[
R_{fc} = \frac{(R_{\text{tot}})(R_{\text{Losses}})}{(R_{\text{Losses}} - R_{\text{tot}})} \quad (3.6)
\]

All experiments were conducted at a constant temperature difference of 45K between heat sink and ambient air. Measuring the losses was achieved by setting fan speed to 0 rpm, where \(Q_{fc} = 0\), and blocking both heat sink ends with insulation to prevent any flow due to natural convection from exiting the heat sink surfaces where forced convection is apparent during normal operation. The forced convection heat transfer coefficient based on inlet temperature difference is calculated using Eq. (3.6) and presented below as:

\[
h_{ITD} = \frac{1}{R_{fc}A_{\text{conv}}} \quad (3.7)
\]

where \(A_{\text{conv}}\) is the forced convective surface area of the heat sink.
3.1.2 Low profile cooling solutions

The previous section presented the experimental arrangement for miniature finned and finless cooling solutions. This section presents the experimental arrangement for analysing the influence of profile scaling of heat sinks on heat transfer performance. Although profile heights are similar in both experimental analyses, this section is entitled low profile cooling solutions for the purpose of differentiating between both experimental discussions. A Sunon B0535ADB2-8 radial flow fan, having footprint area of 1225mm² [35mm x 35mm] and a profile height of 7mm, was used in this analysis of profile reduction on heat transfer performance. In order to quantify the influence of profile height on the combined fan - heat sink cooling solutions, a range of finned heat sinks were designed based on dimensions shown in figure 3.4 a).

![Figure 3.4: Schematic of a) finned and b) finless heat sink designs showing the common dimensions along with the varied parameters H and L. Dimensions in millimeters (mm).](image)

Three profiles were selected based on fin heights of 4mm, 2mm, and 1mm. The largest profile dimension equates to the standard height of the fan outlet. An arbitrary fin spacing of 1.1mm was set which resulted in 16 channels for each finned heat sink examined. Fin thickness was optimised, using Eq. (3.3), to ensure a fin efficiency greater than 99%. In Eq. (3.3), as the heat transfer coefficient and fin height increase, the fin thickness must also
increase to maintain greater than 99% efficiency. Using copper as the heat sink material, the fin thickness must be greater than 0.135mm. The manufacturing technique however required this dimension to be at least 0.3mm resulting in a more than adequate fin thickness for all three designs. In addition to the finned heat sink designs, dimensionally alike finless geometries were comparatively examined with the conventional finned geometries. This design is presented in figure 3.4 b).

A wide range of flow conditions from thermally and hydrodynamically developing to a predominantly fully developed flow pattern within the heat sink channels were achieved by varying heat sink length for the individual profile heights. Hence, as profile height decreases, channel length is increased. Although not optimum for thermal performance in each case, these dimensions were chosen to allow a more complete theoretical validation of the problem by spanning three orders of magnitude of \( L^{*}_{Dh} \). The 4mm profile designs are shown in figure 3.5. The top covers that confine the flow, preventing any bypass during experiments have been removed for illustration purposes to distinguish the heat sink geometries. End views of each cooling solution are presented in figure 3.6, clearly showing the reductions in channel height considered, and also defining the corresponding channel lengths.

As previously mentioned, the fan outlet has a height of 4mm, equaling the largest profile heat sink inlet from figure 3.6. In order to direct all the flow from the fan into the smaller heat sinks of heights 2mm and 1mm, and prevent any bypass effects, diffusers were manufactured to downsize the blower outlet to the appropriate heat sink channel height. Although not required for the heat sinks with a 4mm channel height, a diffuser was also used during experimentation so the developing length between fan blade tip and heat sink entrance was equal for all cases. This diffuser, indicated in figure 3.5, was manufactured from polycarbonate and also acted as an insulator between the heat sink and the die-cast aluminum frame of the fan housing.

The fan and heaters were powered using a TTi dual DC power supply for a range of speeds between 3000 rpm and 11,000 rpm. Fan speed was measured and monitored during experiments using an Omega HHT13 optical tachometer. The heat sources used for each heat sink were Minco thin-film polyimide heaters, attached to the base and sized to closely
CHAPTER 3. EXPERIMENTAL TECHNIQUES AND FACILITIES

Figure 3.5: Finned (left) and finless (right) heat sink designs. Details of the fan outlet diffuser are also shown.

Figure 3.6: Finned (top) and finless (bottom) heat sinks showing reductions in profile height.
match the base footprint area. In doing so, the resistance to heat spreading could be min-
imised. The heat sink temperature was monitored using K-type thermocouples, embedded
in the base and side of the heat sink, and also attached to the top cover of the heat sink.
Another K-type thermocouple was used to record the ambient air temperature. All tem-
perature data was recorded as in section 3.1.1. The experiments were undertaken for an
almost isothermal boundary condition, as a maximum temperature difference between heat
sink walls of approximately 3% was noted. This was slightly lower than that described in
section 3.1.1 as the copper heat sink walls are double the thickness of the miniature heat
sinks which therefore increased heat spreading performance of the low profile heat sink
designs. Embedding of thermocouples into the walls of the miniature heat sink appara-
tus in section 3.1.1 was avoided due to the wall thickness being insufficient. However it
was assumed that this had a negligible influence on the heat transfer measurements as the
thermocouples were firmly mounted to the surface and free from external effects due to the
insulative tape around the heat sink. The heat transfer performance was calculated based on
the expressions provided in section 3.1.1. The experimental non-dimensional heat transfer
performance was then calculated using Eq. (2.21).

3.1.3 Component placement

The experimental setup to determine the influence on heat transfer of positioning a discrete
heat source, at various locations downstream from a radial flow fan is shown in figure
3.7. Two 80mm diameter polycarbonate plates act as a bounding channel for the airflow
exiting the centrally positioned radial flow fan. On the top plate, an orifice was machined
which corresponded to the fan inlet diameter. Two fan diameters of 15mm and 24mm with
a profile height of 4mm were examined for a range of rotational speeds and component
positions, outlined in table 3.1. For the investigation into the positioning of the component
on the top and base plates, a slot was machined in the respective polycarbonate plate which
is detailed in figure 3.8. This channel allowed for the positioning of a copper component,
with dimensions 18mm(L)×8mm(W)×2mm(H), at various radial positions from the fan
blade tip.

In figure 3.8, both top and base plate experimental configurations are shown for a flush
component orientation. In addition, the heat transfer performance of a raised component was analysed and is also shown in figure 3.8. In this set of experiments, the copper component was raised to protrude 1mm above the polycarbonate plate. This protruded arrangement was chosen to replicate electronic components which can typically protrude $\approx 1\text{mm}$ above the surface of printed circuit board designs.

The copper component was heated so a temperature difference of approximately 45K existed between the surface and the ambient surroundings, using a 16mm×6mm Minco thin-film heater. Two K-type thermocouples were used to monitor the component temperature. The first thermocouple was positioned towards the leading edge of the component, and a second thermocouple was located near the trailing edge. An additional K-type thermocouple was used to record the fan inlet and surrounding ambient temperature. All temperature data was recorded as in section 3.1.1. The experiments were undertaken for an
almost isothermal boundary condition, as a maximum temperature difference of 3% was noted between the previously defined temperature measurement locations at the leading and trailing edges. A TTi dual DC power supply was used to power the thin-film heater and also a 16mm diameter Maxon 110049 motor (indicated in figure 3.7) which was used to rotate the radial fan designs.

Thermal performance was expressed in terms of component heat transfer coefficient, independent of secondary heat transfer losses, which is defined in section 3.1.1 by Eq. (3.7). This was achieved by directly measuring the secondary losses through insulating the component surfaces which were subjected to forced convection.

### 3.2 Characterisation of fan performance

Heat transfer measurements alone cannot describe the overall performance of a cooling solution. Knowing the pressure drop across a heat sink and pumping power required to dissipate adequate heat are essential for optimisation and design selection. Flow rates and
pressure drop were measured for each low profile heat sink in section 3.1.2 over the range of fan speeds considered using a test facility designed to BS 848 (1980). In the following chapter on the advanced experimental facilities, flow and pressure performance were also measured for each fan using this test apparatus.

3.2.1 Experimental setup and procedure

A schematic of the experimental test facility is shown in figure 3.9. A number of orifice plates were used to cover a wide range of flow rate and pressure rise characteristics. The test facility, along with the individual orifice plates, were calibrated previously by Grimes et al. (2005) to determine the experimental flow rate relationship with orifice pressure drop.

![Schematic of experimental flow rate and pressure rise test facility.](image)

Figure 3.9: Schematic of experimental flow rate and pressure rise test facility.

In addition to the test facility of figure 3.9, a Furness FC0510 digital micromanometer was required for differential pressure measurement. The positive port on the manometer was connected to a tube which was open to the atmosphere. The negative port was connected to a tube which branched into two, allowing measurement of either differential fan inlet pressure or orifice pressure through on/off valve adjustment.
3.2.2 Data analysis

The relationship between volumetric flow rate and differential pressure across an orifice was used to determine fan performance. These relationships for a range of orifice diameters 5.5mm - 23mm were determined previously by Grimes et al. (2005) and are included in Appendix B. For rotational speeds where fan performance was not measured directly, conventional fan scaling laws from Bleier (1997) were used to scale fan performance. These are defined in Eq. (3.1) and Eq. (3.2).

The performance comparison of dissimilar radial fan designs was presented using non-dimensional flow and pressure coefficients. These coefficients are defined as (Bleier, 1997):

\[
\phi = \frac{\dot{Q}}{\omega D^2 H_f}
\]

(3.8)

\[
\psi = \frac{\Delta P}{\rho \omega^2 D^2}
\]

(3.9)

The importance of blade Reynolds number on fan performance has been discussed in the literature (Quin and Grimes, 2008; Walsh et al., 2009b; Neustein, 1964). As this thesis examines the thermal performance of cooling solutions using miniature fan designs, some scaling effects may become apparent when operating at low Reynolds numbers due to the increased ratio of viscous to momentum forces. It is therefore necessary to define the blade Reynolds number for both fan designs. These are defined in Eq. (3.10) and Eq. (3.11) for axial and radial fan designs.

\[
Re_b = \frac{\rho V_b c_{mid}}{\mu}
\]

(3.10)

where \( V_b \) and \( c_{mid} \) are the blade velocity and chord length at the blade mid radius. For radial fan designs:

\[
Re_c = \frac{\rho V_t c}{\mu}
\]

(3.11)

where \( V_t \) and \( c \) are the blade tip speed and chord length.
3.3 Uncertainty analysis

The adapted form of experimental uncertainty was shown by Kline and McClintock (1953), for a constant odds result, to be found through the use of the root-sum-square combination of the individual contributions from the measurement variables.

\[ \delta W = \left\{ (W_{X1+\delta X1} - W_{X1})^2 + (W_{X2+\delta X2} - W_{X2})^2 + \ldots + (W_{XN+\delta XN} - W_{XN})^2 \right\}^{1/2} \] (3.12)

In Eq. (3.12), each \( W_{X+\delta X} - W_X \) term defines the result as a function of an independent variable. Moffat (1997) developed this technique further into a sequentially perturbed method, also highlighting likely sources of uncertainty in thermal measurements. Using this method of uncertainty analysis, the relative uncertainty in the presented data have been classified using Eq. (3.13). A sample uncertainty calculation using Eq. (3.12) and Eq. (3.13) is provided in Appendix B.

\[ u_W = \frac{\delta W}{W} \] (3.13)

**Bulk heat transfer and fluid measurement uncertainty**

The uncertainty in the measured values of voltage, current, temperature, length, and pressure were estimated as 10mV, 10mA, 0.1K, 50\( \mu \)m, and 5\% of the measured pressure. All measurements recorded were analysed using this approach to determine the largest variance. This resulted in the examination of 36 samples. Consequently, the maximum uncertainties in the forced convection thermal resistance, Nusselt number, volumetric flow rate and \( L_{Dh}^* \) were found to be 5.37\%, 5.44\%, 2.3\% and 2.8\% respectively. The optical tachometer used for measuring fan rotor speed has an accuracy related to the resolution limit of 1 rpm. For the entire speed range considered however, uncertainty in the fan rotational speed was noted as approximately 50 rpm, due to variations in speed monitored over the test duration.
3.4 Closure

The methods for determining bulk heat transfer performance of various finned and finless heat sink designs have been discussed in this chapter. These measurement techniques have also been applied to examine the influence of component placement relative to radial fans. The experimental facility for determining pressure and flow rate characteristics of fans has been presented. The results of the bulk measurements outlined are discussed in Chapter 5. In the following chapter, advanced experimental techniques which facilitate the investigation of local surface heat transfer and velocity field information are presented.
Chapter 4

Advanced experimental techniques

In the previous chapter the experimental methods for determining bulk thermal performance of cooling solutions and components, as well as fan performance have been presented. Cooling solution designs can be further optimised by examining local surface heat transfer and velocity field distributions. An examination of the local heat transfer distribution can highlight regions of increased or decreased heat transfer, hence local heat sources can be positioned accordingly. For complex fluid flow and heat transfer systems that are absent from the literature, it can be difficult to infer why such variations in local heat transfer exist without knowledge of the velocity field which drives the heat transfer. In this chapter, the techniques used to address local heat transfer and velocity field information are presented. These advanced techniques for determining local heat transfer and velocity field information have been developed and assessed in order to obtain accurate quantitative data.

4.1 Local heat transfer measurements

Local heat transfer measurements were acquired for the cases of a) axial fan jet impingement normal to a flat surface and b) radial fan air flows over heated flat surfaces representative of a finless heat sink design. The spatial distribution in heat transfer was investigated by combining infrared thermography and a heated-thin-foil technique. This local heat transfer measurement method has recently received attention at the macro scale and for high
forced convection heat transfer coefficients (>\(10^3\) W/m\(^2\)K). However, there is a need for further investigation into the validity of the technique for miniature scale forced convective applications which are considered in this thesis. Consequently, an assessment of the heated-thin-foil technique, and refinement of the infrared camera calibration, were carried out to characterise experimental errors and uncertainties.

### 4.1.1 Heated-thin-foil technique

The heated-thin-foil technique is analysed using an energy balance method to elucidate the forced convective heat transfer performance independent of secondary heat transfer mechanisms such as natural convection, conduction, and radiation. As the heat generated by Joule effect is dissipated by a number of heat transfer modes, an energy balance based on the axial fan experimental configuration in figure 4.1 provides a method to obtain the local heat transfer coefficient. This conservation of energy is defined in Eq. (4.1) as (Incropera and DeWitt, 1996):

\[
\dot{E}_{in} + \dot{E}_{gen} - \dot{E}_{out} = \dot{E}_{st}
\]  

A differential control volume for the combined paint and foil layers is also included in figure 4.1. A number of assumptions have been made to simplify an energy balance of this control volume. These assumptions are as follows:

- The experiment is carried out in a steady state condition, therefore the storage term, \(\dot{E}_{st}\), is assumed negligible.
- The same temperature field is assumed across the thickness of the thin-foil, as \(Bi < 0.1\).
- The foil and paint layers have the same temperature field due to the low Biot number and negligible contact resistance as shown by Nogueira et al. (2003) and Raghu and Philip (2006). Conduction in the z-direction is therefore negligible and the analysis can be considered two-dimensional.
The last assumption is validated by experimental measurements of acrylic paint thermal properties on stainless steel backings by Nogueira et al. (2003). It was concluded that there was an absence of any defects or air gaps at the interface layers. A similar study by Raghu and Philip (2006) also resulted in the same findings for matt black paint coatings on various materials including stainless steel. Considering both the foil and paint layers in figure 4.1 simultaneously, Eq. (4.1) expands as

\[
\begin{align*}
[q_x]_f + [q_y]_f + [q_x]_p + [q_y]_p + \dot{q}_{gen} dx dy t_f &= [q_x+dx]_f - [q_y+dy]_f - [q_x+dx]_p - [q_y+dy]_p \\
-q_{fc} - q_{nc} - q_r &= 0
\end{align*}
\] (4.2)

where subscripts \( f \) and \( p \) denote the foil and paint layers. The thin-foil is polished stainless...
steel 304 with a low emissivity. The radiation heat transfer from this surface was estimated to be \(< 1\%\) of \(q_{\text{gen}}''\) and therefore negligible. The conduction heat rates perpendicular to the control surfaces at \(x\) and \(y\) can be calculated using Fourier’s Law (Incropera and DeWitt, 1996).

\[
q_x = -k_t t.d. \frac{\partial T_w}{\partial x} \tag{4.3}
\]

\[
q_y = -k_t t.d. \frac{\partial T_w}{\partial y} \tag{4.4}
\]

The conduction heat rate leaving the control area \(t.d.\) is the conduction heat rate at \(x\) including the magnitude it changes with respect to \(x\) over the differential length \(dx\). This is expressed as:

\[
q_{x+d_x} = q_x + \frac{\partial q_x}{\partial x}.dx \tag{4.5}
\]

Similarly, in the \(y\)-direction:

\[
q_{y+d_y} = q_y + \frac{\partial q_y}{\partial y}.dy \tag{4.6}
\]

Substituting Eq. (4.3-4.6) into Eq. (4.2) yields:

\[
(k_f t_f + k_p t_p) \left( \frac{\partial^2 T_w}{\partial x^2} + \frac{\partial^2 T_w}{\partial y^2} \right) dx.dy + \dot{q}_{\text{gen}} dx.dy - q_{f_c}'' dx.dy - q_{w_c}'' dx.dy - q_{r}'' dx.dy = 0 \tag{4.7}
\]

Eq. (4.7) assumes a constant thermal conductivity and thickness for both the foil and paint layers. Generalising and rearranging the terms above to express the forced convection heat transfer rate is shown in Eq. (4.8).

\[
q_{f_c}'' = q_{\text{gen}}'' - q_{w_c}'' - q_{r}'' + \left( k_f t_f + k_p t_p \right) \left( \frac{\partial^2 T_w}{\partial x^2} + \frac{\partial^2 T_w}{\partial y^2} \right) \tag{4.8}
\]

The net conductive heat flux within the foil and paint layer \(q_c''\), is defined by the closing term in Eq. (4.8). Expressions for calculating the remaining terms are available in the
CHAPTER 4. ADVANCED EXPERIMENTAL TECHNIQUES

literature, and shall be discussed in this section also. The thin foil is heated by Joule effect and therefore the generated heat flux is calculated using Eq. (4.9).

\[ q_{\text{gen}}'' = \frac{VI}{A} \]  

Eq. (4.9)

On the observation side, the heated foil represents a flat plate in natural convection. Many relationships for flat plate natural convection, based on orientation of the heated surface and Grashof number exist in the literature (Fujii and Imura, 1972; Radziemska and Lewandowski, 2001; Dayan et al., 2002). An expression for a heated flat plate facing upwards with a constant heat flux boundary condition empirically determined by Fujii and Imura (1972) was utilised for the analysis of the contribution of lateral conduction in the heated-thin-foil technique. This is included in Eq. (4.10).

\[ q_{\text{nc}}'' = \frac{0.13Ra^{\frac{1}{3}}k(T_w - T_\infty)}{A/P} \]  

Eq. (4.10)

The characteristic dimension in Eq. (4.10) is \( A/P \) and \( k \) represents the thermal conductivity of air evaluated at a specified reference dependent on wall and ambient temperatures. It should be noted however that the majority of literature provides bulk or mean heat transfer relationships which may be unsuitable for regions of analysis near the heated-thin-foil edges, as edge effects can no longer be neglected and temperature gradients across the foil will result in a non-uniform heat transfer coefficient due to natural convection. To account for edge effects in experiments which examine the full foil width, empirical relationships were developed and will be discussed in the following section on validating the technique.

The radiation from the high emissivity opaque surface at the observation side to the ambient surroundings can be accounted for using (Incropera and DeWitt, 1996):

\[ q_r'' = \varepsilon\sigma_{sb}(T_w^4 - T_\infty^4) \]  

Eq. (4.11)

where \( \sigma_{sb} = 5.669 \times 10^{-8} \text{W/m}^2\text{K}^4 \) is the Stefan-Boltzmann constant and \( \varepsilon \) is the emissivity coefficient for the matt black surface (\( \varepsilon \approx 0.96 \)). Finally, the forced convection heat transfer coefficient can be evaluated using Eq. (4.8) and the aforementioned expressions in Eq. (4.9)
where \( T_w \) is the measured surface temperature and \( T_{aw} \) is the measured adiabatic wall temperature. Meola and Carlomagno (2004) discussed the distribution of the adiabatic wall temperature along a jet impingement plate for \( 0 \leq M \leq 0.71 \). As the Mach number approached 0, \( T_{aw} \) approached a constant value, regardless of position along the impingement plate, which resembled the ambient temperature. In the current study, \( M \sim \mathcal{O}\left(10^{-3}\right) \) and a comparison of the recorded adiabatic wall temperature and ambient temperature measurements showed that \( T_{aw} = T_\infty \).

The non-dimensional heat transfer distribution is presented in terms of the Nusselt number, defined as:

\[
Nu_L = \frac{h_f c L}{k} \tag{4.13}
\]

where \( L \) is the characteristic dimension, and is dependent on the experimental arrangement examined.

### 4.1.2 Assessment of the heated-thin-foil technique

The uncertainty and validity of the heated-thin-foil technique was investigated by examining two foil thicknesses under identical experimental configurations and convective flow arrangements. The stainless steel 304 foils have a thermal conductivity of 16.3W/m.K and nominal thicknesses of 12.5\( \mu \)m and 50\( \mu \)m with a tolerance of \( \pm 15\% \) provided by the manufacturer. Two axial flow fans were considered for the analysis and are shown in figures 4.2 and 4.3. The Ø24.6mm fan was operated at a speed of 3000 rpm and was positioned 5mm from the foil surface to produce significant temperature gradients. A second flow arrangement was created using a Ø48.5mm fan, also 5mm from the foil surface, but at a speed of 6000 rpm. The Ø24.6mm fan operating at 3000 rpm provides a flow rate of \( 7.487 \times 10^{-5} \)m\(^3\)/s of air at free delivery when the static pressure is zero. The static pressure at the point of no delivery when the flow rate is zero is 4.72Pa. The Ø48.5mm fan operating
at 6000 rpm has corresponding values for the flow rate and static pressure of 0.00568 m$^3$/s and 52.85 Pa, respectively. Both fans provide relatively complex, non-uniform heat transfer characteristics on the foil surface and hence are a good choice for assessing any errors rather than considering a practically unrealistic one-dimensional problem.

In Eq. (4.7), the foil and paint thicknesses can contribute to the conduction in the thin-foil technique; hence, it is an important parameter and was considered in detail. The uncertainty in the foil thickness provided by the manufacturer was negated by measuring the actual thickness for both samples. Taking an energy balance of the heated-thin-foil, the paint thickness on the observation side may result in a contribution to the lateral heat flow; hence, this was also measured. Samples of the painted 12.5 $\mu$m foil and 50 $\mu$m foil used in experimentation were set in a clear epoxy resin. The epoxy block was then ground and polished 2 mm below the edge of the foil sample using a Struers LaboPol-5 variable speed polishing table combined with a semi-automatic LaboForce-3 specimen mover. This
was necessary to find the true foil–paint cross-sectional dimensions as the edge of the foil sample had a burred and distorted cross section along with damage to the paint layer from the sample cutting process. Using a high-resolution CCD camera connected to a Zeiss Axioskop microscope with a 40X lens and a numerical aperture 0.85, images of the foil and paint layers were acquired and sample images are presented in figure 4.4.

The bright white layer is that of the stainless steel foil cross-sectional thicknesses and the darker black layer contiguously above the foil in figure 4.4 a) is the matt black paint cross section. The microscope stage was fitted with a PIMikromove motion controller which allowed X–Y movements of 0.05\( \mu \)m, enabling accurate measurements of both foil thicknesses and paint thickness to be recorded. The average measured foil and paint thicknesses are presented in table 4.1 along with the values stated by the manufacturer.

![Cross-sectional images of the nominal a) 12.5\( \mu \)m thick foil and high emissivity paint, and b) 50\( \mu \)m thickness foil without paint.](image)

**Figure 4.4:** Cross-sectional images of the nominal a) 12.5\( \mu \)m thick foil and high emissivity paint, and b) 50\( \mu \)m thickness foil without paint.

**Table 4.1:** Foil and paint specifications.

<table>
<thead>
<tr>
<th>Description</th>
<th>Manufacturers ( t(\mu m) )</th>
<th>Measured ( t(\mu m) )</th>
<th>( \sigma (\mu m) )</th>
<th>% outside stated</th>
</tr>
</thead>
<tbody>
<tr>
<td>SS 304 foil</td>
<td>12.5</td>
<td>14.3</td>
<td>0.0334</td>
<td>+14.5</td>
</tr>
<tr>
<td>SS 304 foil</td>
<td>50</td>
<td>41.7</td>
<td>0.0455</td>
<td>−16.6</td>
</tr>
<tr>
<td>Matt black paint</td>
<td>-</td>
<td>21.81</td>
<td>5.66</td>
<td>-</td>
</tr>
</tbody>
</table>
The thermal conductivity of the paint used was not available from the manufacturer and as a result a value of 1.38 W/m.K was chosen based on the findings of Raghu and Philip (2006) for matt black paint coatings. Although the paint thickness varies across the foil surface, the mean thickness of 21.81 μm has been assumed in Eq. (4.7) for the entire surface to simplify the analysis. This assumption is shown to be acceptable for the energy balance method in a future section on the validity of the correction method in this chapter. More importantly, due to the magnitude of the stainless steel foil thermal conductivity (16.3 W/m.K), the standard deviation of the foil thickness must be small to prevent large errors in the local heat transfer calculation. As this is < 0.05 μm for both thicknesses considered, and lower than the accuracy of the motion controller, it is sufficient to assume a constant thickness throughout.

4.1.3 Infrared thermography and camera calibration

An in situ procedure, presented by Schulz (2000), was implemented to calibrate the Indigo Merlin Mid camera instrumentation. This enabled accurate temperature measurements of the heated-thin-foil surface. In figure 4.5, a calibration curve relates camera output signal to thermocouple temperature measurements on the surface of the heated-thin-foil. Figure 4.6 indicates the reduction in error through the introduction of this in situ approach. Temperature measurement uncertainty was reduced from 2.8K to 0.2K. Further details of the calibration procedure are provided in section B.1.3 of Appendix B.

![Figure 4.5: Camera calibration fit with measured surface temperature data.](image-url)
4.1.4 Experimental setup and procedure

The details of the experimental test facilities designed for the analysis of axial and radial flow fans are presented within this section. The axial fan jet impingement experimental schematic is shown in figure 4.1. Two fan designs, presented in figure 4.2 and figure 4.3, were chosen to cover both miniature and macro scales. The specifications of the fan designs are given in table 4.2. The heated-thin-foil is representative of a heated flat plate being cooled by an axial flow fan. A stainless steel 304 grade foil with a measured thickness of 14.3μm is clamped and tensioned using copper busbars and a tensioning mechanism that prevents deflection of the flat plate under the impinging forces of the fan air flow. An electric current is passed through the electrically resistive thin-foil, using a TTi TSX1820P DC power supply, resulting in heating of the plate by Joule effect to produce a constant heat generation condition on the surface. Voltage and current were monitored using two Fluke 45 Multimeters. A separate TTi dual DC power supply was used to control fan rotational speed.

The thermal images of the flat plate were acquired using an Indigo Merlin IR camera, calibrated in situ (section 4.1.3), with a 25mm lens providing a standard angular field of view of 22°×16°. For all experiments examining the heat transfer performance of the axial and radial flow fans, this provided a spatial temperature resolution of 312.5μm. For the experiments carried out to validate the heated-thin-foil technique, the introduction of a 6mm extension ring between the lens and camera sensor provided a spatial temperature
resolution of 154\(\mu\)m. The infrared camera was positioned almost normal to the foil surface in observation, with an inclination of 5° from the vertical plane. The influence of the self-reflection of the IR camera was examined by monitoring the temperature distribution along the surface with \(q_{\text{gen}}'' = 0\) at various angles of inclination (5° - 45°) from the vertical plane. As the temperature of the cold image remained unchanged throughout the range considered, self-reflection was deemed negligible on the experimental set-up aided by the poor reflectivity of the matt black surface. The apparatus was positioned in an enclosed environment with an opening for the IR lens. This minimised the contribution that external radiation sources, such as nearby power supplies, made to the reflected radiation offset discussed in the previous section. To ensure accuracy in the IR camera calibration during image recording, a single K-type thermocouple remained mounted to the foil in a location void of large gradients in temperature. A K-type thermocouple was also used to obtain the ambient air temperature and was positioned 200mm upstream of the fan inlet. 60 thermal images were recorded at 1 frame/s once the foil reached steady state. These images were then averaged to reduce noise and time-varying fluctuations in the temperature profile that were a maximum magnitude of 10\(^{-3}\) of the averaged temperature map.

In figure 4.7, the experimental schematic for the radial fan tests is presented. For this series of experiments the local heat transfer distribution is investigated on both top and base surfaces. The effects of a cross flow over the fan inlet on heat transfer trends is also considered. An Edmund Industrial Optics translation stage was used to achieve accurate fan positioning relative to the top and base surfaces. This stage allowed incremental movements of 0.01mm in the vertical direction. In all cases a clearance gap of 0.5mm was set between the base of the fan and base plate, and also between the top of the fan blade and top plate. To accommodate the IR camera for experiments which visualised the base plate heat transfer
distribution, the fan was rotated from above its inlet to provide full optical access as shown in the schematic of figure 4.7 a). Any effects of fan blockage due to the presence of the motor (Maxon 110124 22mm diameter 12VDC) and the positioning stage were alleviated by extending the 3mm diameter input shaft such that the motor to fan inlet distance was 45mm. The additional support of an SKF deep groove ball bearing with 10mm outside diameter was positioned 25mm from the fan inlet to stabilise vibrational effects created by extending the distance between motor and fan. This was interference fitted to a perforated stainless steel 316 grade housing mounted on the positioning stage. Further details of the positioning stage along with the motor and bearing housing are provided in Appendix B.

The bearing housing assembly was designed to prevent fan blockage and also reduce the interaction between the crossing air flow over the inlet and the housing, during the crossflow experiments.

In figure 4.8, the four different radial fan diameters examined are presented. One half of the fan diameters were designed to rotate in a clockwise direction, and the remaining designs rotated in an anti-clockwise direction. This was chosen to investigate surface heat transfer phenomena which could be dependent on the fan rotational direction. The specifications of all radial fans investigated are provided in table 4.3.

An additional experimental configuration to those presented in figure 4.7 was also considered to distinguish the contribution of the crossflow on the top plate heat transfer distribution. This configuration is shown in figure 4.9. The experimental design isolates the heat transferred due to the external crossflow by extending the cooling solution 50mm from the crossflow and thermally insulating the heated top plate. This heated surface represents the cooling solution top plate which is only influenced by the crossing air flow. Pressure and flow rate characteristics of the cooling solution with the extended inlet tube were examined to determine if performance was affected. The fan speeds examined corresponded to those investigated for the configurations in figure 4.7, as any adverse effects from the extended inlet were found to be minor compared to the cases without any tube inlet. Thermal resistance due to forced convection was two orders of magnitude lower than the resistance to conduction in the normal direction through the insulative layer. Therefore, it was assumed that the majority of heat was transferred by convection and the secondary modes described
Figure 4.7: IR experimental setup for radial fan designs and local heat transfer analyses for a) base plate, and b) top plate of a finless heat sink geometry.
in Eq. (4.12).

Various cross flow conditions were considered, including cases with the addition of a flat plate above the fan inlet at a height, $H_{CF}$, shown in figure 4.7. In many large scale and portable electronic devices, the auxiliary air flow can be between the fan inlet and device casing, or between printed circuit board stacks which contain the cooling solutions. Hence, the air flow is confined to a parabolic profile approaching the fan inlet. It was necessary to construct the plate material over the region of interest such that it was transparent in the infrared camera wavelength band, as the plate was in the optical path of the IR camera when viewing the top surface heat transfer distribution (figure 4.7 b)). A window was machined out of a polycarbonate plate which encompassed the region of interest. A thin polyethylene film was then used as the window material to allow optical access without disrupting the cross flow development. There were a number of reasons for this choice of material:
Figure 4.9: Distinguishing between the crossflow and radial fan contributions to the top plate surface heat transfer distribution.

- The transmissivity of polyethylene is relatively high over the 3 – 5µm wavelength, increasing with a decrease in material thickness as shown in figure 4.10 by Tsilingiris (2003). Figure 4.10 also shows that the transmissivity is almost constant over a large range of radiant source temperatures.

- The auxiliary air flow was parallel to the polyethylene window and it was therefore possible to use a tensioned thin film without generating any vibrational or “drumming” effects.

- The distance, \( H_{CF} \), between the window and the heated top surface was sufficient to prevent any degradation of the polyethylene material.

- Polyethylene thin film is inexpensive whereas commercially available infrared transparent crystals can be expensive. As a result, the cost of the experimental facility was reduced significantly.

In the experiments requiring the use of the polyethylene window, the total transmissivity is a product of the transmissivities in the optical path of the infrared detector array.
Figure 4.10: The transmissivity of polyethylene for various material thicknesses and radiant source temperatures, replotted from Tsilingiris (2003).

\[
\tau_{\text{tot}} (\lambda, T_{PE}) = \tau_{\text{atm}} (\lambda) \cdot \tau_{PE} (\lambda, T_{PE}) \cdot \tau_L (\lambda)
\]  

(4.14)

The infrared camera software allows user inputs for transmission and temperature of an additional external optic such as the polyethylene window. The transmissivity of the polyethylene window was estimated as 0.91 by varying the transmission value in the software until the minimum difference in temperature between the thermocouple and IR camera reading was achieved. As shown in Eq. (4.14), the transmissivities can vary with both wavelength and temperature, hence a further calibration as described in section 4.1.3 was necessary to reduce experimental error.

In the following section, a validation of the heated-thin-foil correction method is presented using the axial fan experimental configurations discussed in this section.

### 4.2 Validation of heated-thin-foil correction method

Each mode of heat transfer can have an influential contribution to the overall heat dissipation rate when using miniature scale air flows. The contribution of secondary heat transfer modes was quantified for the local forced convection measurement technique using the assessment procedure outlined previously.
An energy balance technique, as given by Eq. (4.12), was employed for data analysis of recorded infrared measurements. The first objective was to quantify the influence on the convective heat transfer coefficient by neglecting the conductional effects. To achieve this, two foils of thicknesses 14.3µm and 41.7µm were examined under the same flow conditions using two different axial flow fans with specifications provided in section 4.2. The foil surface temperature for both thicknesses using the Ø24.6mm fan is presented in figures 4.11 a) and b). The central peak in temperature is a direct consequence of the fan hub. The remaining temperature distribution is produced by the exit flow angle from the fan blades, and the diversion of air flow around the motor supports at the fan outlet. These supports can be seen in figure 4.2 for the Ø24.6mm fan and figure 4.3 for the Ø48.5mm fan. Comparing figures 4.11 a) and 4.11 b), it is apparent that the temperature profiles differ considerably between the two foil thicknesses. This difference is graphically represented in figure 4.11 c) and is due to non-convective heat transfer effects. Figure 4.11 c) is a temperature map of the resultant differences between both foil thicknesses under identical flow conditions and

![Figure 4.11: Temperature profile of a) the 14.3µm foil and b) the 41.7µm foil using the Ø24.6mm axial fan. c) Temperature difference between a) and b) for identical fan speeds.](image-url)
illustrates the need to account for conduction effects on surface temperature.

The forced convection heat transfer coefficient (Eq. 4.12) for both thicknesses neglecting conduction in the foil and paint are shown in figures 4.12 a) and b). This shows large differences in local heat transfer, particularly at local peaks, as a result of temperature differences between both foils. The relative differences in heat transfer coefficient are quantified in figure 4.12 c), and local deviations up to approximately 90% are apparent. Such deviations confirm that the contribution of conduction in the foil should be accurately assessed.

Figures 4.12 d) and e) provide local heat transfer coefficients for both 14.3\(\mu\)m and 41.7\(\mu\)m foil thicknesses accounting for the contribution of conduction in the foil and paint layers. Both cases show good agreement between local heat transfer results, and local deviations are shown in figure 4.12 f). The maximum deviation in heat transfer coefficient between both cases is approximately 13%. Although outside the uncertainty of the measurements (10.9%), the Gaussian distribution is centred on the mean error at 1.25%, with a standard deviation of 2.26%. Consequently, a high confidence in the accuracy of the result is achieved which is within the uncertainty described previously. The energy balance in Eq. (4.12) is therefore validated, as variation of the foil thickness in the measurement technique resulted in the same heat transfer coefficient with a limited error. This limited error may be attributed to the variation of the paint thickness on the surface of the foil which has been neglected to simplify the analysis process. In the current work, it is acceptable to neglect the deviations in thickness shown in table 4.1 due to its low magnitude thermal conductivity (1.38W/m.K) which was determined by Raghu and Philip (2006).

Comparing the corrected heat transfer coefficient profiles of figures 4.12 d) and e) with those of figures 4.12 a) and b), large errors are observed which reinforce the importance of accounting for the contribution of conduction in the result already noted above. A substantial maximum error of 132% in \(h_{fc}\) is apparent by using the 14.3\(\mu\)m foil and neglecting conduction in the energy balance. Using the thicker 41.7\(\mu\)m foil, errors up to 215% in \(h_{fc}\) are noted when neglecting conduction losses. This is expected, as the thicker foil has a higher magnitude conductive heat flow which, if neglected, will influence the local heat transfer results considerably more than the 14.3\(\mu\)m foil. This is represented in figures 4.13 a) and b) which show the local conductive heat flux of the combined foil and paint for both
Figure 4.12: Forced convection heat transfer coefficient a) using the 14.3μm foil and neglecting conductional effects; b) using the 41.7μm foil and neglecting conductional effects; c) error between a) and b); d) using the 14.3μm foil and accounting for all losses - Eq. (4.12); e) using the 41.7μm foil and accounting for all losses - Eq. (4.12); and f) error between d) and e).
thicknesses examined. The maximum absolute conductive heat fluxes for the 14.3μm and 41.7μm foils, with a paint layer thickness of 21.81μm are 460W/m² and 745W/m², respectively. This indicates that the lateral heat flow can be of the same order of magnitude as the input heat flux (~1000W/m²) and therefore no longer negligible at this scale, a similar finding reached by Patil and Narayanan (2005) for microscale air flows. Even for the higher flow rate test case employing a Ø48.5mm fan, the maximum conductive heat fluxes are 290W/m² and 410W/m² for each thickness. This suggests that there is a reduction in the overall contribution of conduction with increased convective heat transfer coefficient; however, it is still significant for the cases analysed. The corrected forced convection heat transfer coefficient using the Ø24.6mm fan is presented in figures 4.12 d) and e), and shows a low magnitude $h_{fc}$ in the region beneath the fan hub. As the forced convection heat transfer coefficient in this region is <10W/m²K, the majority of heat flow is from secondary heat transfer mechanisms. This region has large temperature gradients and subsequently considerable conductional heat flow in the foil and paint layers away from the fan centre as indicated in figure 4.13.

Figure 4.11 c) also shows that the maximum temperature profile differences between both heated foil examples occur beneath the fan hub, and are up to 6.7K. This is a significant difference as it accounts for 25 - 30% of the driving temperature difference ($T_w - T_{aw}$) for heat transfer in this area. With these factors combined, the natural convection, radiation and

![Figure 4.13: Tangential conduction contribution using the Ø24.6mm axial fan with a) 14.3μm foil and b) 41.7μm foil.](image-url)
conduction heat fluxes change relatively by 25 - 30% between the heated foil thicknesses examined.

Figure 4.14 presents the forced convection heat transfer coefficient using the 48.5mm fan and 14.3\(\mu\)m thick foil. This heat transfer coefficient map provides a contrasting profile to that using the Ø24.6mm fan in figure 4.12 d). Consequently the energy balance method was examined for two very different flow conditions and foil temperature distributions, further confirming the validity and use of Eq. (4.12) as a general correction method in this measurement technique. A deviation of 3% in the forced convection heat transfer coefficient is calculated for the region under the fan hub between tests on both thicknesses and both flow conditions from the Ø24.6mm and Ø48.5mm axial flow fans. This accounts for less than 1% of the total input heat flux.

A circumferential average of the corrected and uncorrected forced convection heat transfer coefficient about the fan centre location is shown in figure 4.15 for the Ø24.6mm axial flow fan. The errors previously described through the use of figure 4.12 are concisely shown in figure 4.15 when neglecting conduction as a contributing factor in the measurement of the forced convection heat transfer performance. Figure 4.15 also describes how large gradients and peaks in heat transfer produce the greatest error if conductional effects are ignored. This is an important consideration if the temperature field being examined produces complex thermal profiles as in the current study. Figure 4.15 also includes the influence of the high emissivity paint coating on the resultant heat transfer coefficient. The

![Figure 4.14: Forced convection heat transfer coefficient using the Ø48.5mm axial fan and 14.3\(\mu\)m foil.](image-url)
contribution of the paint coating in the conductive heat flux profiles in figure 4.13 is negligible for the high heat transfer regions and low temperature gradients; however, in the region under the fan hub the paint layer accounts for up to 5.5% of the total input heat flux for the Ø24.6mm fan at 3000 rpm. This is reflected in the radial average plot in figure 4.15. Under the higher flow rate condition using the Ø48.5mm fan at 6000 rpm, the maximum contribution is again at the low heat transfer, high temperature gradient region under the fan hub, and is 3% of the total input heat flux. Although within the measurement uncertainty, this distinguishes the necessity to include the paint layer in the analysis as flow rates and length scales decrease. Consequently, minimising the paint thickness on the painted foil is beneficial in the experimental measurement technique as it reduces the contribution of tangential conduction losses.

Another objective was to obtain the errors induced by assuming the nominal foil thicknesses and ignoring the paint layer in the energy balance. In figure 4.16, the mean radial heat transfer coefficient for each flow condition is shown when the nominal foil thickness, provided by the manufacturer, is assumed and the heat flow caused by temperature gradients in the paint coating is neglected. The errors produced in the result are exclusive to incorrectly measuring conduction losses. Deviations between both cases are evident, particularly in the regions where temperature peaks exist. This is expected as the greatest variation in net heat flux across the foil occurs due to these peaks, and consequently the error by assuming the incorrect foil thickness is enhanced as local temperature peaks are approached. In the current study, high temperature gradients combined with low heat transfer coefficients towards the fan centre distinctly emphasise the resultant error. A maximum
Figure 4.16: Mean radial heat transfer coefficient using a) nominal foil thickness data provided and neglecting paint coating, and b) using measured foil and paint thicknesses.

An error in the forced convection heat transfer coefficient of approximately 30\% is noted assuming a nominal foil thickness and ignoring the paint coating. The equivalent error for the Ø48.5mm fan at 6000 rpm decreased to a maximum of approximately 17\% due to increased heat transfer coefficients. Hence, the concern in knowing the actual foil and paint thicknesses for the energy balance reduces as flow rates and length scales increase. A radial distribution of the corrected data sets for both flow conditions and foil thicknesses are presented in figure 4.16 to clarify the aforementioned findings. Both corrected data sets for each fan case and foil thickness are superimposed and agree comparatively.

Finally, a generalised assessment on the contribution of tangential conduction in the heated-thin-foil technique for various forced convective heat fluxes and temperature gradients has been considered and is presented in figure 4.17. The data presented are compiled from the findings in the current work and the use of a 12.5\textmu m stainless steel foil as a heat flux sensor. The assessment is based on the case of a thermal spot of heterogeneity. When a fluid motion over the heated foil produces this type of thermal spot or peak on the foil surface, heat transfer is at a local maximum or minimum, heat transfer gradients are large, and consequently the conductive heat flux is at an absolute local maximum. As seen in figure 4.17, the influence of tangential conduction for high convective heat transfer coefficient cases is negligible, even for large temperature gradients. As the convective heat flux is reduced by an order of magnitude however, situations where large temperature gradients occur may require knowledge of the conductive heat flux to acquire accurate results. The
possible errors induced by ignoring the true foil and paint thicknesses at these forced convective heat fluxes are also highlighted, and for the two fan cases in the current study which have been discussed. Using an informed approximation of the forced heat convection flux, it is possible to estimate the expected error \textit{in situ} by a simple one-dimensional calculation of the temperature gradient across a local area where gradients or peaks occur.

\begin{figure}
\centering
\includegraphics[width=\textwidth]{figure4_17.png}
\caption{Contribution of tangential conduction in the heated-thin-foil technique.}
\end{figure}
4.2.1 Refinement of natural convection corrections

The previous section discusses the validity of the conduction corrections applied to resolve the forced convection heat transfer coefficient using the heated-thin-foil technique. Due to large temperature gradients evident in all local heat transfer experiments carried out, tangential conduction provides the largest contribution to the secondary losses. In section 4.1.1, a correlation developed by Fujii and Imura (1972) was implemented to describe the contribution of natural convection on the viewing surface of the flat plate for the tangential conduction analysis. The analysis shown in section 4.2 was considered at the plate centre, where the natural convective heat flux over a horizontally orientated plate is at a minimum. Furthermore, the driving temperature differences for natural convection were relatively small, and any differences between the correlation of Eq. (4.10) and the true losses were negligible for the tangential conduction study. A larger field of view was considered for the local heat transfer investigations on both axial and radial fan types, which utilised the full width of the heated-thin-foil, shown in figure 4.20. This voided the use of Eq. (4.10), as the natural convective heat flux increases significantly towards the plate edges due to a reduction in the boundary layer thickness. The fluid streamlines for the case of a horizontally orientated, heated from above, thin foil were acquired using particle image velocimetry and are presented in figure 4.18 for an average Rayleigh number of $9.7 \times 10^5$. A decrease in fluid density above the heated plate results in the fluid motion presented where entrainment of fluid from both sides of the flat plate is evident. The entrained air from either side merges at the plate centre, forcing a change in fluid direction and the creation of a thermal plume above the plate. An enhanced view of the velocity field above the plate half-width is shown in figure 4.19, also for $\overline{Ra}_w = 9.7 \times 10^5$. The velocity magnitude and direction confirms the boundary layer profile previously discussed and which has been presented by Bejan and Kraus (2003).

Although much previous work has been presented in the literature for horizontal flat plates in natural convection, the majority is in the form of averaged Nusselt number expressions. In addition, these expressions for the average Nusselt number have been shown by Lewandowski (1991) and Radziemska and Lewandowski (2001) to vary in their result by up to 100%, suggesting that the expressions provided are limited for use only on
specific boundary conditions and plate dimensions. In order to refine the accuracy of the natural convection term in Eq. (4.12), independent of location on the plate surface, empirical relationships for the cases of a joule heated horizontal flat plate facing upwards and
downwards were developed over the intended range of Rayleigh numbers. The experiment was designed to simplify the correction procedure for natural convection losses on the camera viewing surface. Although the region of interest was just 80mm×100mm, the tensioned heated-thin-foil dimensions were set as 80mm(W)×240mm(L). This plate aspect ratio (W/L) was sufficiently low for the flat surface heat transfer characteristics to resemble that of a semi-infinite flat surface in the region of interest which was located centrally on the heated-thin-foil, as shown in figure 4.20. The busbar and tensioning mechanisms were also designed to reduce entrainment of air from the plate ends, therefore contributing to the semi-infinite flat plate condition. A two-dimensional profile of the heat transfer coefficient for the heated-thin-foil facing upwards and $Ra_w = 3.89 \times 10^5$ is presented in figure 4.20. As a result of the experiment design attributes, the heat transfer coefficient distribution can be independently expressed in terms of the y-dimension. A suitable choice of characteristic dimension is the plate width, W, as the boundary layer growth is over this length scale due

Figure 4.20: A profile of the surface heat transfer coefficient on the camera viewing side of a heated horizontally orientated plate facing upwards.
to the two-dimensional semi-infinite similarity. The relationship between $Ra_w$ and $Nu_w$ is shown for a heated surface facing upwards in figure 4.21 a) and a heated surface facing downwards in figure 4.21 b). In both cases, Nusselt number scales to the $\frac{1}{5}$ th power of the Rayleigh number, similarly noted by previous authors (Dayan et al., 2002; Lewandowski et al., 2000; Martorell et al., 2003; Lewandowski, 1991; Radziemska and Lewandowski, 2001; Aihara et al., 1972). Both empirical correlations in figure 4.21 express the heat transfer due to natural convection within 8.3% error over the range of Rayleigh numbers examined.

The plate centre and plate edge correspond to $y/W = 0$ and 0.5 respectively. For both orientations an increase in the Nusselt number is noted as the plate edge is approached. The correlation for a flat plate facing upwards was utilised in the axial fan analyses (figure 4.1), and the analyses of the top plate heat transfer using the radial fan (figure 4.7 b)). The correlation for a flat plate facing downwards was used for the analyses of the radial fan and base plate heat transfer shown in figure 4.7 a). Finally, the average Nusselt number is presented in figure 4.22 for both plate orientations including reference expressions from literature.

![Figure 4.21: Scaling of Nusselt number with Rayleigh number for a horizontally orientated heated surface a) facing upwards and b) facing downwards.](image-url)
4.2.2 Turbulence detection and frequency response

The energy balance presented in section 4.1.1 was developed to determine the local heat transfer distribution for a time-averaged, steady state condition. In fan flows such as those examined in this thesis, fluctuations in the fluid velocity exist which, upon interaction with the heated surface, can subsequently produce fluctuations in the surface heat transfer coefficient. The surface heat transfer profile over time can therefore be presented using the analysis in section 4.1.1. It may however, be beneficial in some situations to observe these fluctuations in the surface heat transfer distribution, as it can highlight the influence of certain fluid mechanisms, particularly in the absence of velocity field information. An unsteady state energy balance must be applied to the heated-thin-foil as the storage term ($E_{st}$) can no longer be neglected when examining the instantaneous behaviour of the heat transfer distributions. Hence, an additional term to represent the change in internal energy of the foil and paint layers with time is necessary.

$$q''_c = (k_{ft} + k_{pt}) \left( \frac{\partial^2 T_w}{\partial x^2} + \frac{\partial^2 T_w}{\partial y^2} \right) - \left( \rho_f C_p f t_f + \rho_p C_p p t_p \right) \left( \frac{\partial T_w}{\partial t} \right) \quad (4.15)$$

Similarly to the steady state energy balance, the foil and paint layers are considered as having the same temperature field due to the low Biot number associated with the technique.
Therefore, the total thermal diffusivity in the unsteady state conduction equation above comprises of the sum of the foil and paint thermal diffusivities.

The fluctuations in heat transfer coefficient are presented relative to the time-averaged heat transfer coefficient over the recording time interval $i = 1 : n$

$$\sigma_h = \sqrt{\frac{1}{n-1} \sum_{i=1}^{n} (h_i - h_{fc})^2}{h_{fc}}$$

(4.16)

The normalised maximum and minimum fluctuations in heat transfer coefficient on the flat surface were also considered.

$$\sigma_{h,max} = \max\left\{\frac{h_i - h_{fc}}{h_{fc}}\right\}_{i=1:n}$$

(4.17)

$$\sigma_{h,min} = \min\left\{\frac{h_i - h_{fc}}{h_{fc}}\right\}_{i=1:n}$$

(4.18)

where $h_i$ is the instantaneous heat transfer coefficient, and $h_{fc}$ is the time-averaged heat transfer coefficient.

A map of the local heat transfer coefficient is presented in figure 4.14 for the Ø48.5mm axial fan operating at 6000 rpm and positioned 5mm from the impingement surface. The root mean square fluctuations in the heat transfer coefficient with respect to the mean ($\sigma_h$) for this flow arrangement are shown in figure 4.23 for the two foil thicknesses examined in

![Figure 4.23: Fluctuations in the forced convection heat transfer coefficient using the Ø48.5mm fan with a) 14.3μm foil, and b) 41.7μm foil.](image-url)
the assessment of the heated-thin-foil technique.

Unlike the results for the steady state analysis, the correction procedure for the unsteady state analysis provides dissimilar results for the fluctuating heat transfer coefficient for each foil thickness examined under identical flow conditions. Although the amplitude of the fluctuating heat transfer coefficient differs in some regions, the data for both foil thicknesses presents similar trends. Increased turbulence is experienced directly beneath the fan hub and due to wake shedding from the interaction between the rotating fan blades and the three stationary motor supports (figure 4.3). This results in three pronounced regions of increased fluctuations in heat transfer coefficient observed in figure 4.23. Beneath the fan hub, fluctuations in $h_{fc}$ over the mean value are at a maximum amplitude due to the recirculating effect of the impinging air flow. As the time constant, $\tau_c$, is 56.5% greater for the 41.7$\mu$m thickness foil, the response to high frequency turbulence fluctuations is different. This difference in response to identical flow conditions is shown in figure 4.24 where the fluctuating surface temperature is presented for each foil thickness. In figure 4.24 b), the thicker 41.7$\mu$m foil generally results in a greater attenuation of the surface temperature response. Eq. (4.19) includes the additional heat capacity of the paint layer which further increases the time constant, reducing the frequency response.

$$\tau_c = \frac{\rho_f C_p f f + \rho_p C_p f p}{h_{tot}}$$

(4.19)

where $h_{tot}$ is the total heat transfer coefficient. Therefore assuming a sinusoidal fluctuating

![Figure 4.24: Fluctuations in the surface temperature distribution using a) 14.3$\mu$m and b) 41.7$\mu$m thickness foils.](image)
pattern, the cut-off frequency for the thin foil is:

\[ f_c = \frac{1}{2\pi \tau_c} \]  \hspace{1cm} (4.20)

The cut-off frequency, \( f_c \), is the maximum frequency of the turbulent heat transfer coefficient fluctuations which can be fully restored using the unsteady state correction method outlined. The upper limit of the fluctuating frequency detectible using this technique however, can differ from the cut-off value as noted by Nakamura (2009). To investigate the influence of fluctuations in the heat transfer coefficient on the thin foil temperature map recorded using IR thermography, an analytical model representing the current experimental arrangement was implemented. It was assumed that the heat flow into and out of the control volume (figure 4.1), due to lateral conduction, was constant over time to simplify the model, hence it was disregarded from the energy balance of Eq. (4.21).

\[
\left( \rho_f C_f f_f + \rho_p C_p p \right) \frac{\partial T_w}{\partial t} = q''_{gen} - h_{fc} (T_w - T_\infty) - h_{nc} (T_w - T_\infty) - \varepsilon \sigma \left( T_w^4 - T_\infty^4 \right) \hspace{1cm} (4.21)
\]

In the above relationship, the forced convection heat transfer coefficient was represented by a mean and sinusoidally fluctuating component.

\[ h_{fc} = \overline{h}_{fc} + \Delta h_{fc} \sin(\omega t) \hspace{1cm} (4.22) \]

where \( \omega = 2\pi f \), the angular frequency. The frequency of the fluctuations in \( h_{fc} \) were varied and the resultant temperature response of the thin foil was noted. In figure 4.25 a), an example of a forced convection heat transfer coefficient, with a mean value \( \overline{h}_{fc} = 20\text{W/m}^2\text{K} \) and a fluctuating amplitude 10% of the mean, is shown for two different frequencies of 1Hz and 10Hz respectively. The resultant thin foil temperature response is presented in figure 4.25 b). Although the amplitude of the fluctuating heat transfer coefficient is the same for both frequencies, the amplitude of the temperature fluctuations is dependent on the frequency of the heat transfer fluctuations due to the thermal inertia of the combined foil and paint layers.

The relationship between \( f \) and \( \Delta T_w \) was then generalised through the non-dimensional
Figure 4.25: Frequency and amplitude of a) the fluctuating forced convection heat transfer coefficient and b) the resultant thin foil temperature response for the case of a 14.3\,\mu m SS304 foil and 21.8\,\mu m paint coating.

frequency, \( f^* \), and fluctuating temperature amplitude, \( \Delta T^*_w \), defined by Nakamura (2009) as:

\[
  f^* = \frac{f}{f_c} \tag{4.23}
\]

\[
  \Delta T^*_w = \frac{\Delta T_w}{(T_w - T_\infty)} \frac{h_t}{\Delta h_{fc}} \tag{4.24}
\]

This relationship is presented in figure 4.26.

For \( f^* < 0.1 \), the temperature amplitude approaches a constant value. However, as \( f^* \) increases, the non-dimensional temperature amplitude decreases, ultimately reaching zero. As previously mentioned, the heat transfer coefficient can only be fully restored to the limiting or cutoff frequency of the thin foil. It is also apparent from figure 4.26 that temperature fluctuations produced by higher frequencies than \( f_c \) can still be detected by the infrared camera once these fluctuations have an amplitude greater than the temperature resolution of the camera. In order to define the detectible limit of the measurement technique, the trend presented in figure 4.26 for large \( f^* \) is of interest. As shown in figure 4.26, this can be closely represented by the relation \( \Delta T^*_w \approx \frac{1}{f} \). The detectible limit of the measurement technique, or the maximum frequency of the fluctuations which can be detected,
Figure 4.26: A non-dimensional representation of the influence of fluctuating heat transfer frequency on the resultant temperature amplitude.

occurs when $\Delta T_w = \Delta T_{NETD}/\varepsilon$. Hence, this yields the following relationship to describe the maximum frequency:

$$f_{\text{max}} = \frac{\varepsilon \Delta h f c (\overline{T}_w - T_\infty)}{2\pi (\rho_f C_p f t_f + \rho_p C_p p t_p) \Delta T_{NETD}}$$  \hspace{1cm} (4.25)

The maximum frequency is presented in figure 4.27 for the two foil thicknesses of 14.3µm and 41.7µm examined.

Figure 4.27: Maximum fluctuating frequency detectible using the heated-thin-foil and infrared thermography technique.
4.3 Velocity field measurements

A velocity field analysis was undertaken for two experimental arrangements. Firstly, to determine the optimum diffuser angle in the flow alignment analysis of a miniature fan-heat sink combination, described previously in section 3.1.1. The channel spacing within the heat sink designs was as low as 1.1mm. Furthermore, in order to be confident of even air flow distribution within each channel, a full field analysis of the entire heat sink velocity field was necessary. With these considerations, a single dimensional, invasive measurement technique is unsuitable. Particle image velocimetry was chosen to complete this analysis as it is a full field, non-intrusive velocity field measurement technique.

A velocity field analysis was also undertaken on a radial fan centrally located within a finless heat sink, as seen in section 3.1.3 and 4.1.4, which had a channel spacing of just 5mm. PIV was implemented in this case to examine both the velocity and turbulence data within the heat sink. This was used to understand the heat transfer trends observed for the component placement analysis, and also the local heat transfer analysis which was discussed in the previous sections.

4.3.1 Particle Image Velocimetry

In addition to being a whole field, non-intrusive technique, PIV is also an indirect measurement technique. Particles, with a density matched to the working fluid, are introduced into the flow field homogeneously, without disrupting the original velocity field profile. These tracer particles are illuminated twice in succession, over a time interval \( \Delta t \), in a two-dimensional plane of the flow using a high power light source. In this experimentation, a Nd-YAG laser was used for the particle illumination process. The light scattered by the particles is then recorded on two separate frames using a charged-coupled device (CCD) camera, synchronised with the laser pulses. The recorded images are split into subareas known as interrogation regions. Each interrogation region is small enough to assume a single local displacement vector exists for each particle within the region, i.e. the fluid moves uniformly within the interrogation region. This local displacement vector, \( \Delta x \), is calculated using statistical cross-correlation methods. The local velocity vector can therefore
be resolved, using the known time delay between illuminations, $\Delta t$, for each interrogation region. A local velocity vector field is then created when all interrogation regions are resolved collectively.

Although a high spatial resolution can be acquired using this technique, the temporal resolution is limited. Subsequently, a statistical analysis of the recorded data was necessary to investigate the time-averaged velocity and turbulence statistics. This analysis is included in this section.

### 4.3.2 PIV system

The core components of a PIV system have been described in the previous section. The details of these components are presented in table 4.4 for the two experimental arrangements considered.

<table>
<thead>
<tr>
<th>PIV component</th>
<th>Miniature fan - heat sink flow alignment analysis</th>
<th>Radial fans (with/without cross flow) analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Laser</td>
<td>New Wave Research Minilase III (Nd:YAG)</td>
<td>New Wave Research Solo 200XT (Nd:YAG)</td>
</tr>
<tr>
<td></td>
<td>Wavelength: 532nm</td>
<td>Wavelength: 532nm</td>
</tr>
<tr>
<td></td>
<td>Pulse frequency: 15Hz</td>
<td>Pulse frequency: 15Hz</td>
</tr>
<tr>
<td></td>
<td>Sheet thickness: 1.5mm</td>
<td>Sheet thickness: 1.2mm</td>
</tr>
<tr>
<td>Camera</td>
<td>TSI PowerView Plus CCD</td>
<td>TSI PowerView Plus CCD</td>
</tr>
<tr>
<td></td>
<td>Model 630157</td>
<td>Model 630162</td>
</tr>
<tr>
<td></td>
<td>Resolution: 2MP</td>
<td>Resolution: 11MP</td>
</tr>
<tr>
<td>Synchroniser</td>
<td>TSI Laser pulse synchroniser</td>
<td>TSI Laser pulse synchroniser</td>
</tr>
<tr>
<td></td>
<td>Model 610034</td>
<td>Model 610035</td>
</tr>
<tr>
<td>Tracer particles</td>
<td>JEM ZR24/7 Hazer (Particle disperser unit)</td>
<td>Rosco 1700 fog machine (Particle disperser unit)</td>
</tr>
<tr>
<td></td>
<td>Glycol solution, relative density: 1.05kg/m³ (293K)</td>
<td>Glycol solution</td>
</tr>
<tr>
<td></td>
<td>Particle size: ~1-10μm</td>
<td>Particle size: ~1-10μm</td>
</tr>
<tr>
<td>Software</td>
<td>TSI Insight 3G™ (PIV process)</td>
<td>TSI Insight 3G™ (PIV process)</td>
</tr>
<tr>
<td></td>
<td>Matlab® (Velocity data)</td>
<td>Matlab® (Velocity and Turbulence data)</td>
</tr>
</tbody>
</table>
4.3.3 Experimental setup and procedure

Miniature fan - heat sink flow alignment

In figure 4.28, the experimental apparatus is shown. The cooling solution was contained within a glass walled enclosure of dimensions 400mm (L) × 250mm (W) × 250mm (H). This contained the homogeneously distributed tracer particles for the duration of the recording sequence. The laser sheet was directed parallel to the heat sink base, and at a mid-depth location within the heat sink channels. The camera was positioned perpendicular to the laser sheet plane. However, in order to obtain the optical access required for this technique, the copper top cover of the heat sink, shown in figure 3.3 was replaced with a glass slide. Hence all PIV results were obtained without heat transfer. A range of diffusers were considered with an angle, $\beta$, varying between 25° and 60° with both curved and straight walls.

Figure 4.28: PIV setup showing camera and laser positions with respect to the cooling solution arrangement.
Radial fans with and without crossflow

In section 4.1.4, the experimental apparatus for examining the local heat transfer performance is described for cases with and without a cross flow over the fan inlet. This test facility was also implemented for the velocity field experimentation. In figure 4.29, the example of a radial fan without any cross flow effect is shown to describe the typical experimental layout during PIV experimentation. The cooling solution was contained within a glass walled enclosure of dimensions 600mm (L) × 300mm (W) × 300mm (H). In this enclosure, tracer particles were introduced and illuminated using a Nd:YAG laser in a single plane of interest. The camera was positioned perpendicular to the laser sheet. Before experimentation was carried out for the radial fan with cross flow, the cross flow test facility was characterised using PIV. Cross flow velocity, uniformity and turbulence statistics were monitored during this characterisation and results of which are contained in the following section of this chapter.

Figure 4.29: PIV setup for the analysis of the velocity field within a finless heat sink configuration.
Experimental parameters

A number of parameters were varied to improve accuracy of the PIV data. Such parameters include capture time between image pairs (Δt), interrogation window size, image exposure (to produce a high dynamic range), and image pair count. This data is provided in table 4.5. Due to the wide range of fan sizes examined, a large variation in velocity magnitude existed. Consequently, an initial value for Δt was estimated based on the suggested method by TSI (1999):

\[
\Delta t = \frac{\text{FOV}}{\text{RES} \left( n_p \right)} \left( \frac{n_p}{4} \right) U_{\text{max}}
\]  (4.26)

The estimated value was then adjusted accordingly, so particles within the interrogation window travelled approximately \( \frac{1}{4} \) of the length of the interrogation window. High shear flows occur near the fan exit but can reduce by a substantial amount at relatively short distances downstream from the fan blade. In order to closely adhere to the \( \frac{1}{4} \) rule of Eq. (4.26), and maintain accuracy in low velocity regions, multiple recordings were taken for the same experimental conditions with varying Δt in each case. The median of the processed recordings was taken as the appropriate velocity field. By doing so, inaccurate vectors, produced by Δt which was insufficient or inordinate in some regions of the FOV, were eliminated.

Table 4.5: PIV system parameters for each experimental apparatus.

<table>
<thead>
<tr>
<th>PIV parameter</th>
<th>Miniature fan - heat sink flow alignment analysis</th>
<th>Radial fans (with/without crossflow) analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td>Δt (μs)</td>
<td>35 - 100</td>
<td>8 - 2000</td>
</tr>
<tr>
<td>Interrogation window size (pixels)</td>
<td>32×32</td>
<td>32×32</td>
</tr>
<tr>
<td>Vector spacing (μm)</td>
<td>120</td>
<td>150 - 200</td>
</tr>
<tr>
<td>Image pair count (-)</td>
<td>70</td>
<td>750 - 1000</td>
</tr>
<tr>
<td>Recording frequency (Hz)</td>
<td>14.5</td>
<td>1</td>
</tr>
</tbody>
</table>
4.3.4 Image processing

The recorded image pairs undergo two processing steps using Insight 3G™ software. The initial evaluation step determines the components of velocity in the cartesian coordinates of the field of view under investigation. The second step, or post-process, validates the resultant velocity vector field from the initial processing.

Image evaluation

The raw image pairs are processed using a number of plug-ins. The Nyquist grid engine initiates the process, and generates the tiled interrogation regions with dimensions provided in table 4.5 for the experiments carried out. A Gaussian mask engine conditions the interrogation regions, increasing signal to noise ratio where necessary. The correlation is then carried out using a fast Fourier transform. This produces a correlation map which contains a correlation peak describing the displacement peak caused by the contributions of many particle pairs. The distance between the correlation peak centre and correlation map centre is the mean displacement. Other smaller peaks in this map are described as noise peaks produced by random pairings. A Gaussian peak engine is the final plug-in for the evaluation, providing a threshold to filter spurious vectors.

Image post-processing

Post-processing was used to validate the vector field produced from the intial image evaluation. In some cases reflections due to the interaction between the laser light sheet and the fluid bounding walls produced spurious vectors, or outliers. These outliers are clearly erroneous, but may have avoided some of the correction processes in the initial image evaluation. A local vector validation tool was implemented which assessed a \(3 \times 3\) vector neighbourhood size for a specific user defined velocity threshold, based on the median of the vectors within this neighbourhood. For the same neighbourhood size, a vector conditioning tool was used to fill holes in the vector field which were either created due to the PIV image evaluation or deleting of outliers. Image post-processing was only considered for minor levels of outliers or holes. In cases where outliers or holes were common in the
vector field after image evaluation, the velocity field was disregarded, and the experimental arrangement and parameters were assessed.

4.3.5 Data analysis

Two components of the local velocity field \((u, v, w)\) in the spatial coordinate system \((x, y, z)\) can be resolved from the aforementioned image processing technique for each image pair recorded in succession. Using this data, the following equations were implemented for the presentation of results.

The instantaneous and time-averaged velocity magnitudes are defined as:

\[
U = \sqrt{u^2 + v^2} \quad (4.27)
\]

\[
\overline{U} = \frac{1}{N} \sum_{i=1}^{N} U_i \quad (4.28)
\]

where \(N\) represents the number of instantaneous recordings.

Turbulence data was expressed in terms of turbulence intensity in the component directions

\[
I_x = \frac{\sqrt{\frac{1}{N-1} \sum_{i=1}^{N} (u_i - \overline{u})^2}}{\overline{U}} = \frac{\sqrt{u'^2}}{U} \quad (4.29)
\]

\[
I_y = \frac{\sqrt{\frac{1}{N-1} \sum_{i=1}^{N} (v_i - \overline{v})^2}}{\overline{U}} = \frac{\sqrt{v'^2}}{U} \quad (4.30)
\]

and Reynolds shear stress, which describes the time-averaged stresses induced due to the fluctuations within the fluid. For the presentation of turbulence intensity data on the radial fans, \(\overline{U}\) is defined as the mean fan inlet velocity \((\overline{U}_{in})\), calculated from the measured flow rate.

\[
\tau_{xy} = -\rho \left( \frac{1}{N} \sum_{i=1}^{N} u_i v_i - \left( \frac{1}{N} \sum_{i=1}^{N} u_i \right) \left( \frac{1}{N} \sum_{i=1}^{N} v_i \right) \right) = -\rho u'v' \quad (4.31)
\]
In equations (4.29-4.31), $u'$ and $v'$ represent the fluctuating components of velocity deviating from the mean.

### 4.3.6 Statistical analysis

A statistical analysis was completed on the PIV measurements to ensure a sufficient data sample was considered when capturing the fluid velocity and turbulence information. This was necessary as the recording frequency of the PIV technique, shown in table 4.5, results in random sampling of the velocity field and therefore provides uncorrelated data. Ullum et al. (1998) examined the number of PIV vector maps required to produce acceptable accuracy. It was concluded that a vector map count of approximately 500 was necessary, however the analysis was considered for a single point in the flow field for a relatively even velocity distribution and homogeneous turbulence. The velocity field produced by the rotating fan assemblies in this thesis were unsteady and turbulent. This is reflected in figure 4.30 which presents the instantaneous velocity magnitude, time-averaged velocity magnitude, and Reynolds shear stress generated by rotating a 4mm profile radial fan between two parallel plates, where the x-axis denotes the distance from the fan blades. This was chosen as it contains a vortex within the flow, producing multi-directional fluid movement, and a wide range of velocity magnitudes which are considered in the PIV experimentation. Due to the close proximity with the fan outlet, turbulent fluctuations produced by the fan blade wake are also captured. Hence, a two-dimensional study was conducted to determine a sufficient sampling interval during experimentation.

In figure 4.31 a) - h), the distribution of the velocity magnitude and Reynolds stress with increasing sample size is presented for a complex flow arrangement. The statistical analysis of the turbulence information is presented for Reynolds shear stress. As this is a second order turbulence statistic, the errors ensued from an insufficient number of vector maps are amplified. Hence, its convergence also validates the turbulence intensities $I_x$ and $I_y$.

The streamlines in figure 4.31 indicate the time-averaged vortex using 75 samples is dissimilar, in particular at its core, to the 500 and 1000 sample cases which are alike. The errors in velocity are maximum at the vortex core where the velocity approaches zero, and
Figure 4.30: The a) instantaneous velocity magnitude, b) time-averaged velocity magnitude, and c) Reynolds shear stress for a 1000 sample size.
Figure 4.31: Velocity magnitude and Reynolds shear stress for: a), b) 75 samples; and c), d) 500 samples. Also included is percentage error, $E_{R}$, in velocity magnitude and Reynolds shear stress over a 1000 sample data set for: e), f) 75 samples; and g), h) 500 samples.
also where sharp directional changes in the fluid movement exist. The errors in Reynolds shear stress are particularly evident near the bottom surface where the high velocity air flow begins to detach from the surface due to an adverse pressure gradient.

The probability density function (PDF) is used to present the distribution of the error over a 1000 sample ensemble due to an insufficient sample size. A 1000 sample ensemble was chosen as the limit in this experimentation due to the large storage requirements of the technique. Using an 11MP camera (table 4.4), a 1000 sample recording required 40GB of computational storage. Consequently, the error distribution $E_r$ presented in this section is relative to the 1000 sample time-averaged statistics. Figure 4.32 presents the normal distribution of error in the time-averaged velocity magnitude for the cases of 75 and 500 vector map samples. The 75 sample function is spread over a large error band, indicating that this sample will not capture the time-averaged velocity information sufficiently. In fact, there is only a 95% confidence in the result having an error less than 23.37%. Alternatively, the unity area beneath the 500 sample function is contained between a much smaller error band, where a 95% confidence level in the result having an error less than 9.7% is achieved. By stacking each PDF against vector map sample size, it is possible to monitor the convergence of the result to determine an acceptable number of vector maps. This is shown in figure 4.33 for velocity magnitude, turbulence intensities, and Reynolds stress and a confidence level of 95%. For instances where increases in sample size have little or no effect on error reduction, this suggests that an appropriate number of vector maps has been recorded. This is seen in figure 4.33, as increasing the number of vector maps above $\approx 750$ has a minor influence on the time-averaged velocity magnitude result. For the minimum sample size of 750 recorded for the radial fan with/without crossflow, the velocity magnitude will be within 5.44% error. The turbulence statistics however, appear to require a larger sample size as seen by the lack of convergence to a constant value for the Reynolds stress data.

Although there is quite a significant uncertainty in the Reynolds stress ($>20\%$), the qualitative trends as seen in figure 4.31 are similar, and a trade-off between computer memory requirements and number of samples was necessary. Consequently, no larger than 1000
sample size was recorded during experimentation. In the discussion of velocity field information, the ensemble averaged data is referred as a time-averaged quantity.

In table 4.5, the sample size of the miniature fan - heat sink flow alignment study is provided. As seen in figure 4.31 a) for a 75 sample case, the general flow direction is captured. The primary objective of the PIV experimentation in the miniature fan-heat sink flow alignment study, was to determine the optimum flow alignment angle. Hence, based on the statistical analysis above, a relatively small sample size can be used to obtain this information. For this reason a sample size of 70 was chosen which also conserved computing requirements.

Figure 4.32: Probability density functions for velocity magnitude.

Figure 4.33: Stacked probability density functions indicating 95% convergence criteria for velocity magnitude, turbulence intensity, and Reynolds stress data.
4.4 IR and PIV measurement planes

In figure 4.7 and 4.9, the infrared measurement planes for the radial fan local heat transfer analyses are presented. Local velocity field measurements have been included for the discussion of the local heat transfer distributions. In section 4.3.3, the PIV technique has been described, including an outline of the experimental layout in figure 4.29 for the forward curved radial fan. In this layout, an example of a single measurement plane between the finless base and top plates is shown. A number of measurement planes were considered in the velocity field analysis to investigate the local heat transfer trends observed on the base and top plate surfaces. These measurement planes are labelled in figure 4.34 along with their relative position to the IR measurement planes on the base and top plates. The direction of the crossing air flow is also indicated for the crossflow experiments. Ideally, a fully heated top plate with an inlet orifice centrally located on the heated-thin-foil would determine the full field heat transfer distribution on the top plate. However, initial experiments of this design were unsuitable due to the local variation of the input heat flux $q''_{gen}$ over the surface of the heated-thin-foil. Due to the discontinuity of the fan inlet orifice, electrical current is diverted around the orifice resulting in non-uniform $q''_{gen}$. The non-uniformity of $q''_{gen}$ for this arrangement is described with a temperature distribution of a heated-thin-foil with an orifice in Appendix B. To overcome this, the existing top plate design could be rotated about the fan inlet orifice relative to the crossing air flow. However, for the experiments with a cross flow, the full-field base plate measurements sufficed for the purpose of the cross flow experimentation and the top plate was not rotated about the inlet. This is not necessary for the experiments without a cross flow, as the fan outlet flow was found to be axisymmetric about the axis of fan rotation.

Three measurement planes V1 - V3, as seen in figure 4.34, are positioned 90° apart to examine the velocity field between the finless plates. The inclusion of the crossflow generator prevented velocity field measurements in a fourth plane between V2 and V3, however it was anticipated that the planes V1 - V3 would suffice for the discussion of the heat transfer distribution on the finless surfaces. The plane labelled V4 examines the velocity field at the inlet side of the fan, above the top plate and in-line with V1. The location of this plane was selected on the intuition that the greatest opposition of the cross
flow to the fan inlet flow would occur in this region. A fifth measurement plane, V5, was also considered on the inlet side. This was positioned parallel to the top plate, and 2mm above the fan inlet orifice. The confinement plate was excluded for the measurement of velocity field data in this region.

### 4.5 Crossflow generator characteristics

A crossing air flow was introduced above the radial fan inlet to recreate a practical operating condition and investigate if any variation in the surface heat transfer distribution existed over the standard case without crossflow. A crossflow generator was designed for this requirement, and is shown in figure 4.35 along with the experimental apparatus. The primary objectives were to design a compact generator which provided a uniform velocity over the fan inlet with relatively low turbulence intensity levels.

The design consists of three Ø58mm axial flow fans positioned at the inlet side, a short mixing chamber, a bank of over 900 Ø4.9mm circular ducts, a three-dimensional contraction, wire mesh screen, a second bank of straightening ducts, and a final mesh screen before the outlet side. The first bank of flow straightening ducts were designed to create a fully developed velocity profile upon exiting the individual ducts for the highest volumetric flow.
rate considered. The pressure drop associated with the straightening ducts was calculated (Eq. 2.12), and the auxiliary fans were selected based on the combined pressure-flow rate characteristics of axial fans in parallel. A contraction was then used to produce a uniform outlet flow with reduced turbulence intensity (Winkler et al., 2007). An inlet to outlet area contraction ratio of 4.2:1 was selected with a contraction length ($L/0.5H$) of 0.87. Due to dimensional constraints, the contraction ratio is relatively low when compared to typical large scale wind tunnel designs which can be of order 25:1 (Chong et al., 2009). Such large scale designs can produce turbulence intensity levels less than 0.5%, however for smaller scale designs with lower contraction ratios, this can reach to 5-15% outside the potential core as noted by Winkler et al. (2007) for a contraction ratio of 4.9. As the current compact flow straightener is not in adherence with typical wind tunnel designs, a uniform velocity and relatively low turbulence intensity could not be assumed without a design qualification. This was achieved through local velocity field and surface heat transfer measurements.

Figure 4.36 presents the measurement planes for the characterisation of the crossflow generator. Velocity field measurements were recorded in three planes in the streamwise
In this section, velocity field and surface heat transfer data are presented for an intermediate auxiliary fan speed of 3000rpm and without the presence of a confinement plate. The velocity magnitude and directional vectors are shown for a section of plane B, along the centreline of the crossflow generator outlet, in figure 4.37. The velocity magnitude and direction is uniform and approaches zero at the wall ($z = 0$). Figure 4.37 also includes turbulence intensity levels in the streamwise ($x$) and cross-stream ($z$) directions respectively. In the streamwise direction, the turbulence intensity is generally 5-10% up to 20mm from the wall, with the exception of the boundary layer region at the wall surface where it is <15%. Outside the core flow >30mm, the turbulence intensity increases due to the interaction between the crossing flow and surrounding ambient air. In the cross-stream direction,
Figure 4.37: Velocity magnitude and turbulence intensity in x- and z-directions for plane B without a confinement plate.
turbulence intensity is lower as the fluctuating velocity component in the z-direction is suppressed by the two separate banks of straightening ducts.

In figure 4.38, the variation of the velocity magnitude in the spanwise (y) direction is shown. There is a deviation of ±8%, which is created due to the variance in local velocity between each straightening duct at the exit of the crossflow generator. Although this could possibly be reduced by enlarging the mixing chamber between the auxiliary fans and the straightening ducts, combined with additional mesh screens, this deviation in the velocity magnitude is acceptable for the current analysis. A comparison of the velocity field data between planes A - C confirmed the maximum deviation to be within ±8%.

The local heat transfer coefficient is provided in figure 4.39 a) for plane E. This also indicates the air flow uniformity in the spanwise direction. In figure 4.39 b), a representation of the effects of flow turbulence on surface heat transfer is also shown. At y = 0 and y = 0.1m, turbulence fluctuations induced at the edge of the potential core flow are noted, similar to that seen in the cross-stream direction (figure 4.37). Although the heat transfer coefficient fluctuations are attenuated to some extent (section 4.2.2), the mean fluctuation is just 0.405W/m²K with a standard deviation of 0.183W/m²K over the entire FOV.

Figure 4.38: Uniformity of the velocity magnitude and velocity vectors for plane D without a confinement plate.
Figure 4.39: The local heat transfer coefficient a), fluctuations in the local heat transfer coefficient b), and average local Nusselt number in the streamwise direction c) for plane E without a confinement plate.

For a heated plane E, heat is transferred to the ambient fluid by the crossing air flow which passes over an unheated section on approach (figure 4.36). This is representative of laminar flat plate heat transfer with an unheated starting length that has been theoretically investigated by Bejan and Kraus (2003) and Ameel (1997). In figure 4.39 c) the relationship between local Nusselt and Reynolds number is presented for both experimental data and theory for a constant heat flux surface condition with an unheated starting length. A maximum difference of 13% exists between experimental and theoretical results, further reinforcing both the suitability of the crossflow generator, and also the accuracy of the experimental facility including IR and PIV measurement techniques outlined previously.
Table 4.6: Crossflow velocities.

<table>
<thead>
<tr>
<th>Crossflow generator Aux. fan speed (rpm)</th>
<th>Average velocity, $U_{CF}$ (m/s)</th>
<th>No confinement</th>
<th>$H_{CF} = 26.8$mm</th>
<th>$H_{CF} = 6.7$mm</th>
</tr>
</thead>
<tbody>
<tr>
<td>1500</td>
<td>0.55</td>
<td>0.51</td>
<td>0.21</td>
<td></td>
</tr>
<tr>
<td>3000</td>
<td>1.19</td>
<td>1.14</td>
<td>0.73</td>
<td></td>
</tr>
<tr>
<td>5000</td>
<td>2.06</td>
<td>1.96</td>
<td>1.48</td>
<td></td>
</tr>
</tbody>
</table>

In table 4.6, the crossflow velocities are provided for each experimental configuration and auxiliary fan speed. As discussed in the introductory chapter, the inclusion of a plate above the fan inlet, combined with crossing air flows, is to investigate the heat transfer performance under practical operating conditions for both miniature and low profile cooling solutions in portable and desktop applications. The flow rate of the auxiliary desktop system fan, which produces the crossing air flow above the fan inlet of local cooling solutions, can be constant or adjusted based on the enclosure ambient air temperature. In many desktop applications, printed circuit boards are added as upgrades to a computing system. These upgrades are often much smaller than the enclosure volume, and if forced convection cooled, the hot air is typically exhausted from the enclosure through vents on the casing. Consequently, the ambient enclosure temperature may only rise minimally, resulting in little or no variation in the auxiliary fan speed and flow rate. A constant flow rate was also considered here, as seen in table 4.6. The average crossflow velocity decreases with decreasing confinement height, $H_{CF}$, due to the increased resistance to air flow. As expected, there is a linear relationship between the average velocity and the fan speed of the three auxiliary fans in parallel. A wide range of velocities were considered with a maximum of approximately 2m/s. This range was considered based on typical velocities encountered in desktop computers (Mansingh and Misegades, 1990).

### 4.6 Uncertainty analysis

An uncertainty analysis using the same method as outlined in section 3.3 was undertaken to determine the uncertainty in the local heat transfer and velocity field measurements. The proceeding sections present these uncertainties.
Local heat transfer measurement uncertainty

The uncertainties in the measured values of voltage, current, temperature and foil surface area were estimated as 10mV, 10mA, 0.2K and $1 \times 10^{-6}$m$^2$, respectively. The nonlinear expression in Eq. (4.12), used to calculate the forced convection heat transfer coefficient, was analysed numerically using a finite difference scheme based on the five-point central difference formula. This method produces truncated errors $O(\triangle^2)$ (Croft and Stone, 1989). Therefore, the interval size was chosen to adequately suppress the influence of these errors on the overall heat transfer coefficient. Round-off error, inherent with numerical schemes, was kept to a minimum by implementing a code which calculated the second derivative of the thin-foil temperature using double precision and a suitable number of mathematical operations. Non-uniformities in foil temperature due to variations in the electrical resistance of the material were assumed minimal based on the uniformity of infrared temperature maps ($q''_{fc} = 0$), experiments conducted over a relatively small temperature range 300 – 350 K, and knowledge of the foil thickness and its limited deviation from the mean value (table 4.1). Consequently, the maximum uncertainty in the forced convection heat transfer coefficient was estimated 10.9%. For the local heat transfer measurements, drift in rotational speed was not as noticeable as in the isothermal measurements due to the lower time taken for the thin-foil to reach a steady state condition. Uncertainty in fan speed was noted as approximately 20 rpm, due to variations in speed monitored over the test duration.

Through the introduction of an in situ camera calibration technique, variable but deterministic errors were minimised. This greatly reduced the overall uncertainty due to the potential of varying atmospheric contributions influencing the camera temperature readings.

Velocity field measurement uncertainty

In determining the uncertainty in the PIV experimentation, a combined approach was implemented. This utilised Eq. (3.12) in conjunction with calibration graphs presented by Hallberg (2000), to source the uncertainty in the displacement measurement, $\Delta x$, for a correctly aligned and focused image set. This was estimated as being 2.09% for the miniature
fan - heat sink flow alignment experiments, and 1.86% for the radial fan with and without cross flow investigations. Uncertainties in the timing setup and in the spatial calibration were estimated as 1.4% and 0.4% based on the resolution of the TSI synchroniser and the camera magnification settings. The relative uncertainty in the velocity magnitude determined by the PIV system due to these fixed errors was 2.15% and 1.93% respectively. The temporal uncertainties have been discussed in a previous section on the statistics of the time-averaged quantities presented.

4.7 Closure

The experimental facilities for investigating local heat transfer distributions and velocity field data have been developed for axial fan jet impingement, and radial fan and finless heat sink arrangements. A technique which utilises a heated-thin-foil as a two-dimensional heat flux sensor, combined with infrared technology to provide surface thermographs, has been implemented to investigate local surface heat transfer distributions. The accuracy of this technique for resolving local heat transfer coefficients has been assessed and validated. The limitations of the technique for resolving fluctuations in the heat transfer coefficient have been discussed using analytical and experimental findings. The results of these local heat transfer measurements are presented in Chapter 6 and 7 for axial and radial fan designs.

The experimental arrangements for investigating the velocity field within miniature finned and finless heat sinks, and also radial fans and finless heat sinks, have been discussed. In Chapter 5, these measurements are presented for the miniature finned and finless heat sink designs with the purpose of optimising bulk heat transfer performance. In Chapter 7, velocity and turbulence statistics are presented in the discussion of the surface heat transfer distributions.

Finally, both local heat transfer and velocity field techniques have been used to characterise a compact cross flow generator which has been designed to introduce a crossing air flow over the inlet of radial fans. The influence of this practical arrangement on fluid dynamics and heat transfer in a radial finless heat sink is discussed in Chapter 7.
Chapter 5

Miniature and low profile cooling solutions

Fluid dynamics and heat transfer data are presented in this chapter for conventional finned and novel finless heat sink geometries of miniature dimensions. The thermal performance and flow rates for a range of finned and finless cooling solutions were determined using the experimental methods and facilities detailed in Chapter 3. The velocity field at the mid-plane of both heat sink designs was also obtained using a non-intrusive PIV technique outlined in section 4.3.3. The velocity field analyses have been used to select suitable diffusers between the fan outlet and heat sink inlet which would align the fan outlet flow with the heat sink channels. An image of this arrangement has been shown previously in figure 3.3. These miniature cooling solutions are presented firstly to compliment the limited amount of previous literature on flow alignment in low profile finned heat sinks using radial fans (Walsh et al., 2008). In addition, the influence of flow alignment on the thermal performance of a finless heat sink design is investigated and compared to the typical arrangement of a fan and heat sink without flow alignment as considered by Egan et al. (2009). The thermal performance of both finned and finless designs is also compared for a range of fan rotational speeds using a commercially available radial fan to determine the suitability of a finless heat sink design over the conventional finned design at miniature scales.

This is followed by an investigation into the influence of heat sink profile scaling on
thermal performance of finned and finless designs. A commercially available radial fan was also selected for this purpose, however with different performance characteristics, dimensions, and blade design to assess larger flow rates. A wide range of flow conditions were considered to examine the impact of profile scaling and determine if a cross-over in thermal performance existed between finned and finless designs. The theory presented in Chapter 2 on heat exchanger design has been utilised to compliment the experimental findings by firstly highlighting any deviations in performance predictions when designing fan - heat sink solutions separately. The combined experimental and theoretical analyses have also been used to provide a suitable heat sink design selection tool for thermal designers of low profile forced convection cooling solutions where a radial fan and heat sink are positioned adjacently.

The final section examines a frequent situation in low powered electronics applications where electrical component heat loads can be managed sufficiently by direct component cooling in the absence of a thermally conductive heat sink. A low profile radial fan is positioned between two parallel plates which act as printed circuit boards with an integrated electrical component. The apparatus also geometrically resembles a radial fan and radial finless heat sink. The experimental facility for this arrangement has been discussed in section 3.1.3. The thermal performance of an isothermal component, representing an electronic component, is discussed for various positions from two different radially discharging fans. The importance of component placement is demonstrated with thermal performance and velocity field measurements which highlight the unique fluidic features of the finless design.

5.1 Miniature cooling solutions

The objectives of this work are to provide optimised thermal management solutions based on previous analyses by Egan et al. (2009), using one of the smallest commercially available fans combined with finned and finless heat sink geometries. Optimisation of such solutions has been considered using velocity field and thermal resistance measurements presented in this section. Velocity measurements are considered first in order to develop
a suitable diffuser for each case based on design criterion previously discussed by Walsh et al. (2008) for radial fans and radial finned heat sinks. Thermal resistance and heat transfer coefficient measurements are presented following the flow field analysis, quantifying the influence of the introduction of diffusers on both finned and finless heat sinks over a range of fan rotational speeds.

### 5.1.1 Flow alignment diffuser

Figure 5.1 depicts the flow at the entrance region to the heat sink for both finned and finless geometries without the use of a diffuser from Egan et al. (2009). A range of diffusers were examined for both heat sink geometries producing various heat sink to fan orientations from 25° to 60°. Therefore, due to the quantity of PIV results analysed, the quantitative data presented for this work are only considered for the diffusers, which would optimise flow alignment and distribution in both heat sink cases. Figure 5.2 presents a number of qualitative images of the flow distribution in the diffuser and heat sink entrance region at the medium speed of 5500 rpm. It was found that a 25° curved diffuser for the finless heat sink and a 50° straight diffuser for the finned heat sink were most beneficial.

The left column of images in figure 5.2 illustrates the flow patterns when using similar diffusers with a finned geometry. The images are colour contrasted to distinguish the fins (white) from their shadows in the diffuser (gray). These were unavoidably created by the laser beam used to illuminate the seeding particles within the heat sink. Consequently particle tracking was not possible in these regions of the diffuser. However, this does not affect the validity of the current work as flow structures are still visible in the diffuser, and concentration on flow characteristics within the heat sink is of primary interest for optimising this thermal management solution. A reduction in the angle that flow exits the fan is evident when compared with the finless images and can be attributed to the pressure rise associated with the finned geometry. The white dashed line across the width of the heat sink indicates the entrance to the heat sink from the diffuser. At the lower angle of 25° the flow is relatively evenly distributed among the channels; however, this turning angle is seen to be insufficient in eliminating the impinging flow at the channel entrances against the fins, a similar effect to that shown in figure 5.1 a). The aim of this work is to define
CHAPTER 5. MINIATURE AND LOW PROFILE COOLING SOLUTIONS

Figure 5.1: Normalised velocity magnitude data replotted from Egan et al. (2009), of flow in the entrance region of a) finned heat sink channels 1–6 and b) finless heat sink with no diffuser. All velocities are normalised with respect to the maximum velocity.

the influence of flow alignment with the finned geometry; hence this diffuser is unsuitable for further thermal analysis. A 50° curved diffuser improved flow alignment, however, also resulted in a poor distribution across the heat sink channels. Hence, it is assumed that a 40° diffuser would produce a similar, or worse, scenario than this and was excluded from the PIV analysis. The optimum condition occurs when using a 50° straight diffuser, as both flow alignment and even distribution of flow are achieved. Increasing the diffuser angle to 60° results in an aligned flow with the heat sink channels, however an unbalanced distribution across the channels is noticeable.

The right column of images in figure 5.2 shows the flow distribution in the finless heat sink as a result of varying diffuser angles. It is clear that as the diffuser angle is increased
Figure 5.2: Normalised velocity magnitude plots depicting flow within the diffuser and heat sink entrance for both finned (left) and finless (right) geometries with diffuser angles of 25-60\(^\circ\). All velocities are normalised with respect to the maximum velocity. Regions in gray are shadows from the heat sink fins, which result from the laser light entering from the right.
above 25° the flow does not become evenly distributed within the heat sink. As this pattern becomes more dominant with increased angle, it is expected that no benefits would be achieved in flow alignment with the 60° diffuser. For this reason, PIV analysis was disregarded using this diffuser.

The velocity magnitude data for flow in both types of heat sink where diffusers are introduced to straighten the oncoming flow from a driving fan are presented in figure 5.3 for the finned design and figure 5.4 for the finless design. The velocity results presented in all figures are velocity magnitudes with vector length and contour colour varying accordingly; longer vectors represent high velocity regions. The relationship between colour and velocity magnitude is indicated in the legend provided with each figure.

The results from PIV analysis on the finned heat sink and 50° straight diffuser are presented in figure 5.3. The velocity magnitude plot shown in figure 5.3 presents the fluid flow through the diffuser and heat sink. The diffuser and heat sink are labeled in figure 5.3 a) and also colour contrasted as previously mentioned. For each case considered in figure 5.3, the flow entering the heat sink shows minimal impingement on the fins as opposed to figure 5.1 a), where no diffuser is used. Assuming a fully developed condition at the exit of the heat sink, the maximum and minimum velocities and hence flow rate were recorded in channels 3 and 6 for all three fan speeds. The percentage differences in flow rates between channels 3 and 6 for fan speeds of 3000 rpm, 5500 rpm, and 8000 rpm are 42%, 48%, and 43%, respectively. However, the standard deviation of the flow rates and velocities for all six channels is higher for a fan speed of 8000 rpm, suggesting that the flow distribution between channels is not as uniform at higher speeds. It can be seen from figure 5.3 c) that the angle of the flow entering the diffuser changes at 8000 rpm and that a longer diffuser but of the same angle may be required.

The PIV results for the finless heat sink and 25° curved diffuser are shown in figure 5.4 for the range of fan rotational speeds. Comparing the data presented in figure 5.1 b) for the case without a diffuser and that shown in figure 5.4, it is possible to distinguish noticeable differences in the flow distribution due to the introduction of a diffuser. In figure 5.4 the flow distribution is relatively even in the channel of the finless heat sink and all speeds examined resulted in a similar evenly distributed flow pattern. The majority of the channel
surface area is therefore influenced by the bulk exit flow of the fan, unlike in figure 5.1 b) for the case without a diffuser where a large region of low fluid velocity exists near the right wall at the entrance.

Figure 5.3: Velocity magnitude plots of flow in a finned heat sink and 50° straight diffuser for fan speeds of a) 3000 rpm, b) 5500 rpm, and c) 8000 rpm. Respective scale bars for velocities in m/s are given above each result. Flow is from left to right. Regions in gray are shadows from the heat sink fins, which result from the laser light entering from the right.
Figure 5.4: Velocity magnitude plots depicting the flow within a finless heat sink and 25° diffuser for fan speeds of a) 3000 rpm, b) 5500 rpm, and c) 8000 rpm. Flow is from left to right. The dashed line represents the entrance to the heat sink from the diffuser.
5.1.2 Thermal performance

Following the selection of a diffuser for the finned and finless heat sinks using the flow field analyses discussed, experimentation was arranged to obtain thermal resistance measurements for a range of fan speeds from 0 rpm to 8000 rpm. The results are plotted in figure 5.5. This figure, along with all subsequent figures discussed, also compares data recorded by Egan et al. (2009) for the finned and finless cases where no diffuser was used. From figure 5.5 it can be seen that a reduction in thermal resistance can be obtained through the use of a diffuser. This confirms the findings of Walsh et al. (2008) who examined the benefit of flow alignment in radial finned heat sink designs. As the rotational speed of the fan increases, the heat sink and diffuser combination outperforms the heat sink without a diffuser. For a finless heat sink, a reduction in thermal resistance is achieved when using a diffuser between approximately 4000 rpm and 8000 rpm. Below this range of speeds, more significant reductions in thermal resistance are seen by using a diffuser to align the flow exiting the fan. In order to achieve a forced convective thermal resistance of lower than 115K/W using a finned heat sink and diffuser combination, the fan must operate at a rotational speed greater than 5000 rpm. Alternatively, the use of a finless heat sink and diffuser combination produces similar thermal resistances from a fan speed of just 3000 rpm. A benefit of this is that lower operating speeds are sufficient for similar cooling effects when using the finless arrangement. In terms of fan reliability, noise, and pumping power this proves highly advantageous. The secondary cooling mechanisms observed in the experimental arrangements are also shown in figure 5.5 and were found to give a thermal resistance, $R_{Losses}$, of 63K/W.

For a fan speed of 8000 rpm, using the diffuser designed for the finned heat sink, a decrease of approximately 23% in forced convection thermal resistance is obtained. At the same operating speed, the finless heat sink and combined diffuser provide a decrease of 15.3% over a finless heat sink without diffuser. It is therefore apparent that in the case of a fan and heat sink in parallel, aligning the flow with the heat sink channels produces lower thermal resistances than impinging flows. At 8000 rpm the finned diffuser case results in a lower thermal resistance compared with the finless-diffuser solution. This is an
expected result as the design of the finned heat sink was optimised for 8000 rpm, as discussed in section 3.1.1. As previously mentioned, a difference in velocity magnitudes for both geometries can be seen in figures 5.3 and 5.4. This can be accredited to a higher system resistance due to the finned heat sink, which implies that higher mass flow rates are evident in the finless heat sink for each rotational speed recorded. Higher mass flow rates commonly result in increased heat transfer rates; however, figure 5.5 enforces the influence of boundary layer analysis used in the finned heat sink design, i.e., the optimum spacing of fins in a heat sink for a given pressure drop is determined based on the boundary layers merging just at the exit of the heat sink (Bejan and Sciubba, 1992). Thermal resistance measurements for the finned case drop below that of the finless case at approximately 7500 rpm. However, the thermal resistance values for the finless diffuser case are lower compared with the finned heat sink without diffuser arrangement. Even at 8000 rpm where the forced convective thermal resistance appears to be reaching a constant value with increased rotational speed, there is a reduction of 11.3% over the forced convective thermal resistance of the finned heat sink without a diffuser. This is significant considering that the 25° diffuser used for the finless heat sink only increases the overall footprint area of the thermal solution by approximately 6%. It was also noted that the 50° diffuser used to optimise the finned heat sink increases the overall footprint area of the thermal solution by
approximately 16%. This result demonstrates the need to design integrated and optimised fan and heat sink solutions for the low profile market.

Similarly for a combined fan and heat sink cooling solution, sizing issues may also be associated with the power supply necessary for operation. Considering the power requirements of a cooling solution, it is reasonable to measure shaft power supplied to the rotor when characterising fan performance. This provides designers with power requirements for the fan alone, which may deviate considerably from the motor power required especially when considering small scale fans. The current work is not examined in this manner, as there is an enclosed motor and fan assembly designed for combined use. Power requirements are therefore presented for the motor and fan collectively. The manufacturer provides a nominal operating speed of 6000 rpm. To achieve this rotational speed, a power input of 0.163W is required. This value increases to 0.320W to reach an operating speed of 8000 rpm. Power is therefore conserved to a greater extent when operating the cooling solution below the nominal operating speed of 6000 rpm as opposed to above this speed where power requirements rise sharply for small increases in rotational speed ($P \sim \omega^3$).

Hence, depending on the level of restrictions evident with power supply choice and the thermal performance requirements, it may be beneficial to integrate a finless heat sink and diffuser design as it outperforms the finned heat sink cases below 6000 rpm, shown in figure 5.5. It is also of interest to note that this level of power consumption could be supplied by a high end mobile phone battery such as the Nokia BP-4L battery for a period of 34 hours of continuous use. In reality, however, the fan would only be in use while the device is in high processing mode so the actual lifetime is likely to be limited by phone functionality rather than by cooling solution.

In figure 5.6, the heat transfer coefficient is presented for forced convection cooling. Similar to the thermal resistance measurements, this data is based on the average heat transfer coefficient of the heat sink. The forced convection heat transfer coefficient for the finless cases is almost twice the average heat transfer coefficient compared with the finned cases over the fan speeds tested. This is inversely proportional to the convective surface area of both heat sinks, as given in Eq. (3.7). With reference to figure 5.5, which shows thermal resistance plotted against fan speed, it is possible to conclude that in general over
the range of speeds examined, the finless design will dissipate a similar and, in some cases, favourable magnitude of heat per unit of temperature compared with the finned design. The heat transfer coefficient quantifies this relative to the convective surface area of the heat sink. The finless heat sink considered here has a convective surface area of 44.7% less than the convective surface area of the finned heat sink. It is therefore expected that the heat transfer coefficient values will be greater for the finless design as it dissipates a similar magnitude of heat using almost half the convective surface area of the finned design. Increasing rotational speed results in an increasing difference in heat transfer coefficient between heat sinks using diffusers and heat sinks without diffusers. At 8000 rpm, the finned heat sink achieves an increase of approximately 27% through the introduction of the selected diffuser. For the same rotor speed, an increase of approximately 15% is achieved for the finless heat sink using the diffuser to align the flow. This is due to the increase in flow rate and combined reduction in pressure across both heat sinks through flow alignment. The larger increase in the heat transfer coefficient for the finned heat sink with diffuser emphasises the benefits that can be achieved in finned designs through flow alignment at this scale and possibly at larger scales.

This section has shown how flow alignment can enhance thermal performance of both finned and finless heat sink designs for relatively minor increases in footprint area. It
was also shown how the finless design can be a viable solution in thermal management at miniature scales. The heat transfer measurements however, indicated a cross-over to the finned geometry at the design speed of 8000 rpm. In the following section, the conditions for cross-over between finned and finless geometries is further investigated by examining a wide range of thermal solutions which have been scaled down from a channel profile height of 4mm to 1mm.

5.2 Low profile cooling solutions

The main objective in this section is to quantify the influence on heat transfer due to profile scaling of finned and finless forced convection heat sinks and define the parameters where a finless design can be a beneficial design choice. Optimisation at the low profile scale has been considered experimentally using thermal resistance and flow rate measurements and is presented in this section. Additionally, a useful design optimisation tool, using a laminar duct flow heat transfer model developed by Muzychka and Yovanovich (2004) and heat exchanger theory, is compared with the experimental results and provide a predictive technique for the thermal designer. The theoretical methods for performance prediction of heat exchangers have been described in Chapter 2.

Pressure and flow rate characteristics are presented first, followed by thermal resistance measurements of each heat sink design investigated. Finally, optimisation and geometry choice are deciphered with the associated dimensionless parameters to allow a generalisation of the finned versus finless approach.

5.2.1 Pressure and flow rate characteristics

The pressure and flow rate data used to characterise the fan at the nominal operating speed of 7000 rpm is presented in figure 5.7. The manufacturer’s supplied data (Sunon, 1998) is shown along with an experimentally measured performance curve for the same rotational speed, acquired using a test facility discussed in section 3.2. Fan performance was measured across the range of speeds examined from 3000 rpm to 11,000 rpm. The manufacturer’s stated performance at 7000 rpm was scaled using conventional fan scaling laws
CHAPTER 5. MINIATURE AND LOW PROFILE COOLING SOLUTIONS

Figure 5.7: Fan performance curve at fan speed of 7000 rpm.

(Eq. 3.1 and 3.2) to acquire the performance directly available to the designer over this range also.

In figure 5.7, the supplied data predicts a fan performance of approximately 10-15% greater than that measured for the same fan speed. In the following results, both the experimentally measured and supplied data are utilised to predict heat transfer performance, highlighting any discrepancy when using the available data provided by the manufacturer.

The performance curves depicting mass flow rate through the heat sink, and pressure drop across the same are presented in figure 5.8 for each profile examined. Measured fan characteristic curves for a range of speeds are also displayed, and expressed in terms of mass flow rate. The pressure drop across each heat sink system at each operating point is predicted to within 10% over the entire speed range except for the 4mm profile finless heat sink where no significant pressure drop is registered experimentally. This is due to the lack of resistance in the heat sink geometry as it attributes the same channel dimensions as the fan outlet dimensions and only 17mm in length.

For the majority of heat sinks tested, pressure drop is over predicted with exception being for the 2mm profile finless heat sink which under predicts for fan speeds above 5000 rpm, to a maximum 6% deviation. This over prediction in pressure drop influences the expected mass flow rates considerably for high system resistance heat sinks such as the
1mm finned heat sink where a 10% deviation in pressure drop between the prediction and experimental data leads to a reduction in the predicted mass flow rate of almost 50%.

Due to the contracting effects of reduction in fan outlet from 4mm down to 1mm in height, it was initially expected that the experimental performance curves would show a higher pressure drop than the predicted model as the additional drop in pressure due to this contraction is not included in Eq. (2.12). As previously discussed, the difference in pressure drop between predicted and experimental is almost the same for all examples, including the 4mm profile finned heat sink, without contraction at the fan outlet. Consequently, the influence of this contraction appears to be negligible relative to the overall pressure drop across the heat sink.

The difference in predicted and measured flow rate for the finned heat sinks is displayed in figure 5.9. The data for each profile considered has been normalised with the respective maximum measured flow rates at 11,000 rpm. Two predictions have been compared with the actual mass flow rate measured through each heat sink. One prediction defines the operating points when using Eq. (2.12) and the measured fan performance curves. The other defines the predicted operating points using Eq. (2.12) and the manufacturer supplied
fan performance data, which is directly available to the thermal designer.

As previously mentioned with reference to figure 5.8, as system resistance increases, the deviation between the expected flow rate and the measured data also increases. This is clearly distinguished in figure 5.9 as both predictions for the 4mm profile fit the measured data with relatively minor differences. As profile height decreases and channel length increases however, the under prediction of mass flow rate increases, a pattern that is also noted for the finless cases but at a lower magnitude due to the low system resistance of the finless design. The deviation between both prediction models is never above 13% of the maximum flow rate for each profile, and is closer to 5% for the majority of fan speeds. This difference is caused from the over prediction of fan performance by the manufacturer, described previously through figure 5.7.

5.2.2 Thermal performance

The effect of fan rotational speed on the forced convection thermal resistance is presented in figure 5.10 for heat sink profile heights of 4mm, 2mm, and 1mm respectively. For the 4mm profile, the finned heat sink clearly outperforms the finless alternative over the entire speed range. However, figure 5.10 a) also shows that as fan speed decreases, the performance trend of the finless heat sink is increasing relative to the finned design. At 11,000 rpm the
finless heat sink has a 53% higher thermal resistance but decreases to approximately 33% at the minimum speed of 3000 rpm considered.

The thermal performance of the 2mm profile height finned and finless heat sinks are also presented in figure 5.10. Through the change of profile height and heat sink length, a different trend has emerged. At the lower speeds between 3000 - 7000 rpm, the finless heat sink outperforms the finned comparison by up to 45%. At approximately 7600 rpm however, this increase diminishes and there is a cross over in geometry selection. The use of the finned heat sink produces marginally lower thermal resistance for the remainder of speeds considered. This indicates the importance of the relevant design parameters on heat sink choice, such as fan performance over the intended operating range and dimensional
The benefits of using the finless design is shown when operating in the 3000 - 7000 rpm range. In order to achieve the same finless thermal resistance, at a given speed, using the conventional finned geometry, an increase of approximately 1000 rpm is required. This increases both the pumping power requirement and noise, while also decreasing the fan or blower reliability.

The influence of a further decrease to a 1mm profile height provides a similar conclusion. The lowest thermal resistance achieved at 11,000 rpm using the finned design can be equaled at approximately 7000 rpm using the finless design. An important design consideration particularly in portable electronic devices is power usage. The fan required 0.31W to operate at 7000 rpm and 0.86W at 11,000 rpm. Therefore the combined finless cooling solution can achieve 9K/W at 35% of the power input required to achieve the same using the combined finned cooling solution at this scale.

Figure 5.11 presents the Nusselt number based on the inlet temperature difference, Eq. (2.21), which provides the heat transfer data of figure 5.10 in a concise form. The hydraulic diameter is chosen as the characteristic length scale as it results in each heat sink aspect ratio collapsing predominantly to one line, with a small spread as flow transitions from the developing to fully developed condition. This can be described simply using two asymptotes to denote these flow conditions. Heat transfer in the developing flow region can be described using a model presented by Sparrow (1955) for laminar forced convection in the entrance region of flat rectangular ducts. This was selected as it best fits the experimental data over the transitional region. The fully developed asymptote is formed from an energy balance in the limit of very long ducts, where the fluid outlet temperature asymptotically approaches the wall temperature for an isothermal wall condition. Combining both asymptotes using the Churchil and Usagi (1972) asymptotic correlation method gives:

\[
N_{ITD} \approx \left[ \left( 4L_{Dh}^* \right)^3 \left( \frac{1}{0.664 \sqrt{L_{Dh}^* Pr^{1/6}}} \left( 1 + 7.3 \left( L_{Dh}^* Pr \right)^{1/2} \right)^{1/2} \right) \right]^{-1/3}
\]

(5.1)

The asymptotic correlation of Eq. (5.1) indicates a transition to the fully developed flow condition at \( L_{Dh}^* \approx 0.1 \). The experimental data show good agreement over the range of \( L_{Dh}^* \).
CHAPTER 5. MINIATURE AND LOW PROFILE COOLING SOLUTIONS

considered. However, for the 4mm and 2mm profile finless heat sinks as $L_{Dh}^*$ decreases the deviation in the experimental Nusselt number increases over the expected result. Conversely, as $L_{Dh}^*$ increases for the 1mm finless profile, higher Nusselt numbers were measured compared to those predicted. This is indicative of a delayed onset to fully developed flow which results in higher heat transfer. This suggests that flow patterns created due to the combination of this fan and a larger flow area in the finless design are enhancing heat transfer. Examining Eq. (5.1), it can be shown that as $L_{Dh}^*$ approaches zero, the limiting Nusselt number can be expressed by the relationship for laminar flat plate heat transfer, $Nu = 0.664Re^{1/2}Pr^{1/3}$, as the duct geometry has a minor effect on $Nu$ at this $L_{Dh}^*$. This asymptotic relationship is also presented in figure 5.11, however for a Reynolds number exponent of $\frac{3}{5}$. The experimental measurements are closely correlated using this scaling relationship, confirming the unsteady flow patterns within the finless heat sink design. The larger flow area of the finless design doesn’t confine the flow to the longitudinal direction as rapidly as the finned channels. This has been shown previously in the velocity field measurements of section 5.1. Within the heat sink, the fan exit flow is redirected longitudinally much sooner for a finned geometry than in a finless geometry. Consequently the flow pattern in the finned channels achieves a structured flow similar to a laminar parabolic profile before the heat sink exit. This provides reason for the agreement in figure 5.11 between the finned heat sinks and theory. It is therefore difficult to accurately predict the finless performance as it is highly dependent on fan exit flow characteristics, reinforcing the need to develop both fan and heat sink as a combined cooling solution rather than separately as is largely the case. In the current fan - heat sink arrangement, the empirical correlation of Eq. (5.2) provides a better fit to the experimental data for greater accuracy in the prediction of heat transfer performance.

$$Nu_{TTD} \approx \left[ \left( 4L_{Dh}^* \right)^3 + \left( \frac{L_{Dh}^*}{0.664} \right)^{3/5} Pr^{1/5} \right]^{-1/3}$$  (5.2)

The ratio of heat transfer between dimensionally alike finned and finless geometries can be used to determine a suitable design and is represented in Eq. (5.3). The heat dissipated, $Q$ is based on the experimental conditions of a constant temperature difference of 45K between
heat sink and inlet air temperatures.

\[ Q^* = \frac{Q_{\text{finned}}}{Q_{\text{finless}}} \]  
\hspace{1cm} (5.3)

For example, when \( Q^* < 1 \), the finless heat sink provides a higher heat transfer rate than the finned heat sink design, and should be the preferred choice for the designer. \( Q^* \) can therefore quantify the relative enhancements and reductions in heat transfer performance for both designs. In figure 5.9, the deviation between two mass flow rate predictions, one using Eq. (2.12) and the measured fan performance, and the other using Eq. (2.12) and the scaled manufacturer stated fan performance is presented. The resultant information using the latter of these two predictions for both finned and finless geometries is used to calculate one of the heat transfer predictions labeled in figure 5.12. The other heat transfer prediction in figure 5.12 is calculated using the measured flow rate through the heat sink, described by the experimental data points in figure 5.8. Also included is \( Q_{\text{max}}^* \), the maximum theoretical \( Q^* \) for each fan speed, or flow rate, using the manufacturer’s fan performance data. This was achieved by determining the optimum finned heat sink design which maximised heat transfer performance for an unrestricted fin thickness and fin spacing, where \( n \approx HW_{ch}/ab \). The data is presented against fan rotational speed to deduce the dependency of heat transfer enhancements on rotor speed as opposed to fluid pumping
Although system performance, Eq. (2.12), is described in terms of pumping power, fan performance is relative to fan rotational speed, as shown in Eq. (3.1) and (3.2), and a more suitable choice to compare the attributes of the finned and finless heat transfer ratio.

Bejan and Sciubba (1992) show that optimisation of parallel plate spacing occurs when the plate length is the same order of magnitude as the thermal entrance length. This occurs at $L_{Dh}^* \approx 0.1$. The experimental transition from finless geometry to finned geometry where $Q^* = 1$ in figure 5.12 b) occurs at this point. Extrapolating the 1mm profile data in figure 5.12 c), this transition also occurs at a similar $L_{Dh}^*$ position. The 4mm profile case cannot be considered in the same manner as the finless heat sink Nusselt number is at much lower $L_{Dh}^*$ values than the transitional region, and subsequently is not optimised for any fan speed considered. The 4mm finned and 1mm finless heat sinks have been shown to outperform

Figure 5.12: Ratio of heat transfer rates for heat sinks with a) 4mm, b) 2mm, and c) 1mm profiles.
their alternate geometries for the range of fan speeds examined. The range of $L^*_{Dh}$ for both is similar and is spread across the transitional region, $0.02 \leq L^*_{Dh} \leq 0.2$, from developing to fully developed thermal boundary layers. This region also contains the 2mm profile results where the finned heat sink is beneficial, and where the finless heat sink is the preferred design choice. Consequently, increased thermal performance can be achieved over a large range of fan speeds by operating in the transitional region stated.

For all profiles examined the experimental ratio indicates that a cross over at $Q^* = 1$, from finless to finned design will be incorrectly predicted using the measured flow rates. The predicted cross-over point based on the measured mass flow rates through the heat sink is premature compared to the true cross-over point determined experimentally as in figure 5.12 b) and c). This reinforces the conclusions from figure 5.11 that unsteady flow patterns promoted within the larger flow area finless heat sink due to the air delivery from the blower, are in fact enhancing the heat transfer rates of the finless design above that predicted using the theoretical approach. The extended data in figure 5.12 a) for the 4mm profile design prediction displays a cross over that is almost the same when compared to the experimental heat transfer measurements. However, the remainder of fan speeds considered confirms the ratio of heat transfer is over estimated with both predictions based on measured flow rates and the supplied fan data. The predicted and experimental cross over is clearly shown in figure 5.12 b) for the 2mm profile case. Measured heat transfer rates show that below 7600 rpm the finless design will start to outperform the finned heat sink. The value predicted using the experimental flow rate data however is much lower at 4400 rpm, suggesting that in the design of forced convection cooling solutions at this scale, the performance attributes of the finless design can be under estimated. The same is apparent when comparing the prediction model using the manufacturer’s data available to the designer. In this case a cross over fan speed of $\approx 5000$ rpm is anticipated. The nominal operating speed stated by the fan manufacturer is 7000 rpm. The predicted heat transfer ratio using the manufacturer’s data in figure 5.12 b) suggests that by operating at this speed the finned design outperforms the finless design by 25%, a margin which could clearly distinguish the choice of geometry to the designer. In reality however, the finless design is at a similar level of heat dissipation.
The heat sink length also varies according to profile height to provide a wide range of flow conditions. This influences the difference in experimental and predicted ratios in heat transfer. In figure 5.12 a), over the range of speeds examined for the 4mm profile heat sinks, there is a mean difference of 47% between the predicted ratio using experimental flow data and experimental ratio. However, for the 2mm profile height presented in figure 5.12 b), a pattern begins to emerge that relates the fan rotational speed with the deviation in predicted and experimental data. As fan speed increases, this deviation begins to increase from 38% at 3000 rpm up to 45% at 11,000 rpm. Considering the 1mm profile height, the information based on experimentation presented in figure 5.12 c) produces almost parallel offset curves, and as a result the deviation between the predicted ratio and experimental ratio varies as a percentage relative to fan speed. At 3000 rpm, the enhancement produced in the finless design is much greater than predicted, resulting in a deviation of 36%. This difference begins to reduce almost linearly above 5000 rpm to approximately 15% at the highest speed. The decrease in flow area for both geometries, combined with an increase in length over the previous profile examples, allows the flow to reach a more stable developed form before exiting the heat sink. Hence, a closer representation of the flow profile used in the development of the prediction model. The prediction using the theoretical model with the manufacturer’s data is much lower than that predicted using the measured flow rates. This is related to the system performance of the 1mm profile designs in figure 5.8. The relatively small discrepancy in pressure difference between theory and measured, results in a large discrepancy in flow rate. Consequently, the lower capacitance rate envisaged theoretically produces a lower heat transfer rate. A similar discrepancy between theoretical and measured pressure difference for the 4mm profile in figure 5.8 have a lesser influence on the predicted capacitance rate. Hence, both predicted heat transfer rates are similar as shown in figure 5.12 a). The geometry selection can also be decided based on the above information. As previously mentioned, the theoretical model under predicts the heat transfer for the finless heat sink, however the dimensionless thermal entry length will be predicted close to the actual values based on the lower system resistance curves of the finless designs in figure 5.8. The designer can therefore create a suitable finless geometry to fall in the transitional $L^*_Dh$ knowing that it will outperform an alternate finned design of like dimensions.
A useful design tool given the dimensional constraints in many portable devices combined with the simplicity of the finless design. The adverse of course applies if a finned design is optimised as seen for the 4mm profile in this study, however for low profile portable devices the simple finless concept can prove effective in removing adequate heat with a lower manufacturing cost, reduced weight and reduced fouling issues.

Sections 5.1 and 5.2 present heat transfer optimisation criteria for finned and finless heat sink designs positioned adjacent to an enclosed fan arrangement. A primary objective of this thesis is to optimise forced convection cooling solution designs which can conserve space for low profile devices. This has been achieved in the previous sections for low profile cooling solutions which integrate heat sinks to promote heat dissipation. In the following section, direct component cooling is investigated by rotating a radial fan in the absence of a volute between two parallel plates which could represent adjacent printed circuit boards in an electronic device. This experimental arrangement, shown in figure 3.8, also doubles as an investigation into local heat transfer trends which exist in a radial fan - finless heat sink cooling solution design with radial discharge of air.

## 5.3 Direct component cooling

The previous sections of this chapter examined cooling solution designs which integrated miniature and low profile heat sinks that increased convective surface area. As low profile and portable devices reach the reliable limit of passive cooling, the introduction of a forced air flow may increase the heat transfer coefficient to an acceptable level such that the surface area of the local heat source alone is sufficient for dissipating the necessary amount of heat to maintain reliable component temperatures. In this case, the benefits include reducing cost and cooling solution dimensions. This section discusses the influence of component positioning, relative to a radial fan, on the mean component heat transfer performance. This was achieved through mean heat transfer coefficient measurements on a 18mm×8mm component, also complimented with velocity field measurements to understand the observed trend in component heat transfer. Scaled Ø15mm and Ø24mm radial
fan designs were implemented for this investigation. These fan designs were confined between two parallel plates with a 5mm spacing which could represent printed circuit board surfaces. In addition, heat transfer from flush and raised component orientations are briefly discussed.

5.3.1 Component placement

Firstly, the velocity field between the parallel plates is presented over a distance, $X$, between 0 and 0.04m from the fan blade and at fan speeds of 4300 rpm and 7500 rpm. $X_o$ represents the distance from the fan blades to the leading edge of the component in the heat transfer measurement as shown in figure 3.8. The velocity fields are included for the discussion of the trends in forced convective heat transfer, with varying component position, as a direct consequence of the fan exit flow pattern. Therefore, although buoyancy assisting or opposing flow may be apparent to some extent at the larger $X_o$ positions from the fan blades and lower fan rotational speeds, it was decided to conduct the velocity field analyses in the absence of a heated component to determine if the fan flow field had a noticeable effect on the component thermal performance.

In both figure 5.13 a) and b) for the Ø15mm fan, a high velocity, shearing air flow is experienced along the base surface for $0 < X < 0.007m$ approximately. A vortex pair is generated upon exiting the rotating fan blades, however it is suppressed to the radial direction by

![Figure 5.13: Time-averaged velocity magnitude and streamlines between top and base plates for a Ø15mm fan with rotational speeds a) 4300 rpm and b) 7500 rpm.](image)
CHAPTER 5. MINIATURE AND LOW PROFILE COOLING SOLUTIONS

the inclusion of the top plate. Consequently, these time-averaged vortices, observed near the exit region of the radial fan, result in the fluid forming a jet-like structure which impinges the top plate at $X \approx 0.008\text{m}$. Further downstream, the influence of this impingement effect is experienced on the base plate. The momentum of the impinging air flow results in a deflection from the top plate towards the base plate, which is further aided by the velocity of the second vortex. At $X \approx 0.02\text{m}$, a second impingement with a much lower approach velocity is noted on the base plate.

Figure 5.14 presents the time-averaged velocity magnitude for the Ø24mm fan at 4300 rpm and 7500 rpm. Compared to the velocity field of the Ø15mm fan, a similar vortex structure is apparent, along with a high shearing effect on the base surface, in this case for approximately $0 < X < 0.01\text{m}$. The secondary vortex observed using the Ø15mm fan does not exist at these rotational speeds when using a 24mm fan. This difference is due to the increased flow rate from the Ø24mm fan through the same plate spacing which overcomes the adverse pressure gradient created by the counter rotating secondary vortex in the radial direction.

In figure 5.15, the average heat transfer coefficient of the component is presented for various distances between the fan blade and the component leading edge. For both fan diameters and rotational speeds examined, there is a relationship between the velocity field and component heat transfer performance. In figure 5.15 a), the heat transfer coefficient

![Figure 5.14: Time-averaged velocity magnitude and streamlines between top and base plates for a Ø24mm fan with rotational speeds a) 4300 rpm and b) 7500 rpm.](image-url)
Figure 5.15: Component heat transfer coefficient for varying positions, $X_o$, for a) a Ø15mm fan and b) a Ø24mm fan.

for a component positioned on the base plate is at a maximum at the closest distance to the fan ($X_o = 0.00075m$). A large negative slope in the heat transfer coefficient exists from this maximum up to $X_o \approx 0.005 - 0.007m$. As previously mentioned, figure 5.13 indicates a high velocity, shearing air flow, over $0 < X < 0.007m$. This almost covers the entire surface of the component at $X_o = 0.00075m$. Therefore, as $X_o$ increases, a greater convective surface area of the component becomes unaffected by this high shear flow. This quickly degrades the component heat transfer coefficient, which eventually levels out for $0.005 < X_o < 0.015m$. In this region, the secondary vortex shown in figure 5.13 provides the mode of heat dissipation. The second impingement, which occurs on the base plate, results in an increase in the component heat transfer coefficient at $X_o = 0.01875m$.

The component heat transfer performance, when positioned on the top plate, differs considerably due to the non-uniform velocity distribution created by the radial fan between these two surfaces. Here, the peak heat transfer coefficient is lower in magnitude than the peak noted when positioned on the base plate, and it is also situated 0.00275m from the fan blades. Interestingly, at this position the component is being primarily cooled by the first vortex and also the impinging flow. Contrary to the region $0 < X_o < 0.007m$ for component positioning on the base plate, the heat transfer coefficient for top plate component positioning in this region only varies by a maximum of 20%, with the former placement
varying the heat transfer coefficient by up to 45%. The air which is recirculated by the first vortex is constantly renewed with air at ambient temperature. It is both the first vortex and initial impingement region on the top surface which promotes the heat transfer coefficient above that noted when positioning the component on the base plate in the region $0.00275 < X_o < 0.00975$ m. Therefore, this highlights the advantages on thermal performance when positioning components in a region of impinging air flow. Beyond this position, the air deflects from the top surface to the base surface, and the heat transfer for a component positioned on the top plate decreases below that of a base mounted component.

In figure 5.15 b), the component heat transfer coefficient using the Ø24mm fan is presented. Positioned on the base, the heat transfer coefficient gradually decreases with increasing $X_o$. On the top plate, positioning the component in the vortex and initial impingement region provides an increase in heat transfer over base plate positioning, similar to that observed using the Ø15mm fan.

In theory, with a radially exiting, uniform air flow from the fan, the component represents a heated surface with an unheated starting length, $X_o$, which has been considered in the literature (Bejan and Kraus, 2003; Ameel, 1997). Based on the measured flow rate of the fan, the relationship between the local Reynolds number and local heat transfer coefficient was determined for a laminar and isothermal surface condition. This is presented in figure 5.15 for both fans and a rotational speed of 7500 rpm by integrating the theoretical local heat transfer coefficient over the component width ($W$) to determine the average component heat transfer coefficient ($\bar{h}_{fc}$).

$$\bar{h}_{fc} = \frac{1}{W} \int_{X_o}^{X_o+W} 0.332 \frac{k}{\nu} Pr^{1/3} Re^{1/2} \left[ \frac{1}{X} \right] \left[ \frac{1}{X} \right]^{-1/3} dX.$$ 

For both fan diameters examined, this relationship under predicts the heat transfer performance considerably. This is expected, as the velocity profile between the top and base plates (figure 5.13 and 5.14) is much more complex and non-uniform. The investigation of the two separate speeds of 4300 rpm and 7500 rpm also indicates the same trend in heat transfer is reflected at different flow rates which was found to scale with $Re^{0.6} - Re^{0.7}$ for both fan designs over the entire $X_o$ range. This is shown in figure 5.15 b) which presents the theoretical component heat transfer coefficient for 7500 rpm with a Reynolds number exponent of 0.6 versus the original exponent of 0.5 for laminar flat plate heat transfer. The relationship closely follows the experimental data for the base plate component positioning using
a Ø24mm fan. This is an interesting finding, as it reaffirms the discussion on finless heat sinks in the previous section. Firstly, it shows that the performance of finless heat sink designs can be predicted quite well by increasing the exponent of the Reynolds number closer to that of a turbulent flow regime. Secondly, these modified predictions, shown here and in Eq. (5.2) agree quite well for two different radial fan designs and finless heat sink arrangements.

5.3.2 Flush and raised component orientation

In electronic packages, components can be mounted flush or raised on the printed circuit board. The significance of either orientation on the component heat transfer performance has been considered by examining a component positioned on the base plate for \(X_0 = 0.00075\)m and on the top plate for \(X_0 = 0.00075\)m and \(0.00475\)m respectively. In each case, a range of fan speeds were investigated using the Ø15mm fan. Comparisons of average component heat transfer coefficient for both orientations are shown in figure 5.16.

At the base position, adjacent to the fan exit, a flush mounted component appears favourable. Although the raised component has a greater surface area for convection, it is anticipated that the fluid is forced to detach from the top and side surfaces of the component as it acts as a forward-facing step in the flow field (Abu-Mulaweh, 2005). Furthermore, the trailing vertical surface of the component is not influenced by forced convection, as reattachment of the velocity field is downstream of the component similar to that observed for a backward-facing step in a flow field (Abu-Mulaweh, 2005; Le et al., 1997). This is shown in figure 5.17 for a sectioned view of the component cooling arrangement, and results in lower heat dissipation than the flush orientation, which leaves the flow field undisturbed in this high shear region. For a component mounted on either of the two positions on the top plate, the heat transfer coefficient, in most cases, is the same or better for a component with a raised orientation. As discussed previously, the velocity profile which drives the heat transfer in this region differs considerably than on the base. As the fluid is either recirculated or impinged onto the component surfaces in these locations, the effects of detachment are not significant, unlike for the radial flow over the base plate adjacent to the fan exit. This also agrees with the literature, as air jet impingement of a protruding pedestal results
CHAPTER 5. MINIATURE AND LOW PROFILE COOLING SOLUTIONS

Figure 5.16: Component heat transfer coefficient and relative enhancement for flush and raised orientations.

Figure 5.17: The hypothesised air flow interaction for a raised component positioned on the base plate (section view).
in an increase in heat transfer over that for a flat surface (Fleischer and Nejad, 2004). The level of enhancement for a flush component over a raised component at each position is also provided in figure 5.16.

5.4 Closure

This chapter focused on the possibility of using a finless designed heat sink over conventional finned designs in miniature and low profile applications. Enhancements in thermal performance of both heat sink designs were achieved when the air flow exiting the fan outlet was aligned with the single channel finless and multi-channel finned geometries. This increase in thermal performance was acquired for relatively minor increases in the over all footprint area of the cooling solutions. Aside from the manufacturing cost benefits of the finless heat sink design, increased heat dissipation was noted over a dimensionally alike finned design for a wide range of fan rotational speeds.

In order to determine a cross-over point between finned and finless geometries, low profile designs were examined for various length to height combinations that covered a wide range of flow conditions. This cross-over criteria in design choice for optimum thermal performance corresponds to a design where the thermal boundary layers merge on exiting the heat sink. A design choice can therefore be made based on this criteria combined with other factors such as manufacturing cost, weight, and heat sink fouling. The benefits of a finless design are therefore apparent, as it can dissipate heat at a similar level to the finned design in low profile applications, while also reducing the additional design factors mentioned.

The data presented on component placement serves two purposes. Primarily, it has been shown to distinguish the significance of placement relative to fan induced air flows on the overall component heat transfer coefficient. In addition to this, although the data is an average over an 8mm radial length, it highlights the local heat transfer trend in the radial direction, were the top and base plates to act as a finless radial heat sink. The integral of this radial heat transfer coefficient profile for top and base surfaces over-estimates the overall thermal performance of a radial heat sink of this design, due to the unheated starting
length effect which would not occur if the plates were in an isothermal or isoflux state. However, it does indicate that this cooling solution arrangement could provide substantial increases in thermal performance over commonly implemented enclosed volute designs, which may suppress the fluid structures generated by rotating fans. The arrangement of this cooling solution design is also preferable for lowering footprint areas in dimensionally restricted applications, as the fan is contained within the heat sink, unlike adjacent fan-heat sink cooling solutions also outlined in this chapter. Hence, a further investigation into the velocity field and local heat transfer performance of this type of cooling solution arrangement is considered in Chapter 7. An analogy was also noted between increases in component heat transfer and local impinging air flows over the surface. In Chapter 6, heat transfer performance due to air flow impinging a flat surface is considered using axial fan designs with low profile restrictions.
Chapter 6

Heat transfer with Axial fans

The benefits of flow alignment within heat sink channels on heat transfer have been dis-
cussed in the previous chapter. In addition, conditions where surface impingement occurs
have been briefly shown to provide local increases in the thermal performance of a dis-
cretely positioned component. Further investigation was therefore considered to examine
the thermal performance of a cooling solution which relies on forced convection heat trans-
fer that is predominately driven by an impinging air flow with low profile constraints. To
address this, heat transfer due to axial fan jets normally impinging a flat surface is investi-
gated in this chapter. Local heat transfer measurements are discussed for a miniature scale
axial fan design, which could be implemented in portable applications. Exit flow charac-
teristics of the miniature fan jet were measured to explain the physics of the heat transfer
distribution on a flat surface. A macro scale fan design, typically used in desktop sized
electronic applications, is also considered to determine if similar heat transfer phenomena
exist for macro scales. Finally, important design considerations for miniature axial fans
used in applications of heat transfer augmentation, are outlined to conclude this chapter.

6.1 Axial fan jet impingement

A schematic of the experimental arrangement is provided in figure 4.1 and specifications
of both fan designs are given in table 4.2. Pressure and flow rate characteristics of both
miniature and macro scale axial fan designs are presented first, also showing the influence
on fan performance of a flat plate positioned downstream of the fan outlet flow. This is followed by an investigation of the local heat transfer distribution over the impingement surface with varying parameters of fan to plate distance and fan rotational speed. The measurement of local heat transfer coefficients on the surface was achieved using the infrared and heated-thin-foil apparatus developed in section 4.1, and validated for use with complex fan induced flows in section 4.2. The resultant local heat transfer profiles were then averaged over an area representing an electronic component located at various positions on the impingement surface to assess the influence of component placement on thermal performance. Relatively simple methodologies are outlined in this section for determining the exit flow characteristics of the axial fan jet which is then related to the heat transfer trends observed on the flat surface. Finally, circumferential averages of the surface heat transfer data are used to define the radial distribution in heat transfer and determine if a common scaling relationship with Reynolds number exists for both miniature and macro scale fan designs examined.

6.1.1 Pressure and flow rate characteristics

Fan performance was characterised experimentally using a test facility designed in accordance with BS 848 (1980) presented in section 3.2. Measured static pressure and flow rate data for a Ø24.6mm (figure 4.2) and Ø48.5mm (figure 4.3) axial fans are presented in figure 6.1. The corresponding operating points with the inclusion of a flat plate downstream are also shown for each rotational speed examined. For the Ø24.6mm fan, this range of speeds was 3000 - 12,000 rpm corresponding to a blade Reynolds number, defined in Eq. (3.10), of 2000 < \( Re_b \) < 8000. The larger Ø48.5mm fan was examined over a range of rotational speeds 2000 - 6000 rpm and blade Reynolds number 2800 < \( Re_b \) < 8500.

Quin and Grimes (2008) observed a sensitivity of Reynolds number on fan performance for \( Re_b \) < 1980. Below this Reynolds number the increased ratio of viscous to momentum forces begins to adversely affect fan performance. Fan performance was shown to progressively reduce with further reductions in Reynolds number, and conventional scaling laws (Bleier, 1997) were no longer satisfied. Similarly, Neustein (1964) also noted a Reynolds number effect for \( Re_b \) < 2000. A similar Reynolds number effect was noted for miniature
centrifugal fan designs by Walsh et al. (2009b). As the minimum $Re_b$ for the Ø24.6mm fan approaches this critical value, the effect of Reynolds number on heat transfer performance may also be seen.

In figure 6.1 a), a trend of high pressure low flow rate is observed for the Ø24.6mm fan due to the relatively large hub-tip ratio and correspondingly low exit flow area. The abrupt change in the pressure and flow rate characteristics is the point of stall for this fan. For $H = 2.5\text{mm}$, the largest system impedance is evident, and each operating point occurs outside the recommended operating range of this fan and in the stalling region. Consequently, this fan to plate distance was not considered in the heat transfer analysis. Plate heights of 10mm and 15mm from the fan outlet plane impose minor resistance to the Ø24.6mm axial fan and can be considered as free delivery. The larger Ø48.5mm fan was also examined for the same fan to plate distances with results presented in figure 6.1 b).

As shown in figure 6.1, the flow rate changes with fan distance from the plate for a constant blade Reynolds number, $Re_b$. Consequently, the definition of a suitable Reynolds number to relate non-dimensional fluid flow and heat transfer characteristics was chosen as:

$$Re_d = \frac{\rho Ud}{\mu}$$  \hspace{1cm} (6.1)
where $\bar{U}$ is the mean velocity at the fan exit, calculated using the measured flow rate, $\dot{Q}$. The Reynolds number is defined in Eq. (6.1) based on the characteristic length ($d = D - d_h$), or the slot width of the annular outlet flow leaving the fan outlet. This characteristic length scale was chosen as it represents the hydraulic diameter of the annular outlet flow which impinges the flat surface. Although fan to plate distance influences fluid flow and heat transfer, it is this length scale which primarily governs both the area subjected to forced convection cooling and the outlet flow dynamics of the annular jet. This provided a range $60 < Re_d < 300$ using the miniature Ø24.6mm fan and an order of magnitude increase for the Ø48.5mm fan with $600 < Re_d < 3000$.

6.1.2 Local surface heat transfer

Figure 6.2 presents local heat transfer coefficient distributions on a heated flat plate due to the impinging air flow of axial flow fans. The left column of figure 6.2 presents the surface heat transfer distributions for the miniature Ø24.6mm axial fan at fan to plate distances of 5 - 15mm and a rotational speed of 9000 rpm. The right column of figure 6.2 illustrates the heat transfer distributions for equivalent fan to plate distances using a larger scale Ø48.5mm axial fan at a rotational speed of 4000 rpm. The corresponding $H/D$ are also labeled in the figure caption. In both cases, the fan outlet geometry is superimposed on the local heat transfer maps to understand the foundation of the complex heat transfer pattern. With reference to figure 4.2 and figure 4.3, the location of outlet flow regions, motor supports, and fan hub are all discernible.

The local heat transfer coefficient maps of figure 6.2 distinguish the complex patterns which emerge as a result of the three dimensionality of the impinging air flow combined with a flow interaction with the motor supports positioned on the fan exit flow plane. Six regions of increased heat transfer are evident which can be attributed to the fan blade-motor support interaction. The effect of fan blade-motor support interactions have been investigated in axial fan designs, however mainly in relation to acoustic emissions (Lu et al., 2007; Huang and Wang, 2005). As the fan blades pass over a motor support, the fluid from the pressure side of the blade is forced to divert around the support obstruction. Consequently, streamlines converge at either side of the support, resulting in local maxima in the exit air
Figure 6.2: Forced convection heat transfer coefficient for a Ø24.6mm axial fan at 9000 rpm (left, a-c)) and a Ø48.5mm axial fan at 4000 rpm (right, d-f)) and various fan to plate distances. Ø24.6mm fan to plate distances: a) 5mm ($H/D = 0.203$), b) 10mm ($H/D = 0.407$), and c) 15mm ($H/D = 0.610$). Ø48.5mm fan to plate distances: d) 5mm ($H/D = 0.103$), e) 10mm ($H/D = 0.206$), and f) 15mm ($H/D = 0.309$). Contour level: 5W/m$^2$K.
velocity approaching the heated plate further downstream. Hennissen et al. (1995) also observed this local rise in velocity magnitude when analysing the exit flow profile of a standalone axial flow fan with three motor supports on the exit flow plane. As expected, a sudden drop in exit velocity to zero was noted directly beneath the motor supports. This is reflected in the heat transfer measurements as regions of decreased heat transfer coefficient which resemble the motor support geometry appear on the impingement surface. These regions are at an angular offset to the position of the motor supports upstream of the flat plate. This is caused by the swirl angle of the exit air flow, as the offset is in the fan rotational direction and increases with increasing $H/D$. Similarly, the low air velocity beneath the fan hub also results in a deterioration of the heat transfer coefficient.

Furthermore, the positions of the six peaks in heat transfer coefficient on the flat plate are also influenced by the swirling air flow. As $H/D$ is increased for the miniature fan, the peaks move outwards in the radial direction, while also moving in the direction of fan rotation, $\omega$. This positional change is due to the swirling and expanding air flow leaving the fan outlet, confirmed in a following section on the exit flow characteristics of the fan jet. The discrete regions of enhancement in heat transfer also become easily defined as they are further isolated from each other due to the expanding air flow downstream of the axial fan. At $H/D = 0.203$, the heat transfer gradients are significant, as indicated by the 5W/m²K contour levels in figure 6.2 a). These gradients in heat transfer coefficient decrease as $H/D$ increases, due to the expansion of the outlet air flow downstream, and mixing of the high velocity gradients which originate at the exit flow plane of the axial fan.

For the larger scale fan, the six peaks are also influenced by the swirling outlet flow of the axial fan, however there are some differences with the results for the miniature fan design. The position of three peaks do not change substantially over the fan to plate distances examined which is possibly due to the smaller span of $H/D$ considered. In particular, the three peaks that are situated directly beneath the motor supports are at almost equal locations, independent of $H/D$. As the motor support thickness in the axial direction is approximately one third of the width of the largest motor support, the air flow leaving the fan blade may be deflected off the supports towards the axial direction. Such a deflection
would reduce the swirl angle of the exit flow in this local region. This hypothesis is illustrated in figure 6.3. Consequently, the location of these peaks remains unchanged for the range of $H/D$ considered. This is reinforced when analysing the change in position of the remaining three peaks in heat transfer, which are located beneath the outlet flow area of the fan. These peak locations are no longer independent of $H/D$ as the absolute velocity entails a greater swirl angle.

A numerical study on the acoustic emissions due to fan blade-motor support interaction for circular motor supports by Lu et al. (2007) supports this hypothesis. Pressure contours around the motor support indicate a stagnation point on the support surface corresponding to the near side (NS) of the motor support as shown in figure 6.3. This suggests that fluid impingement and therefore deflection exists in this region. Beneath the support, the referenced authors show a reduction in pressure, which is similar to that observed in the study of bodies subjected to a cross flow. The shear layer separates from the body and vortex shedding occurs resulting in unsteadiness in the near wake region (Ozgoren, 2006; Lankadasu and Vengadesan, 2008).

In figure 6.2 d) - e), the largest peaks in heat transfer coefficient generally occur beneath the three motor supports, as opposed to directly beneath the outlet flow areas. In fact, for both fan designs and all fan to plate distances examined, the maximum peak in forced convection heat transfer is a consequence of the blade-support interaction for the largest motor support which carries electric wiring to the motor. Through these findings, benefits in positioning of discrete heat sources for maximum heat transfer can therefore be attained.

![Figure 6.3: Section view of an axial fan above a flat plate with hypothesised interaction of the absolute velocity field with a motor support.](image-url)
CHAPTER 6. HEAT TRANSFER WITH AXIAL FANS

and will be discussed in the following section for the miniature fan example.

Overall, the region directly beneath the fan hub provides the lowest heat transfer performance with local heat transfer coefficients as low as $6\text{W/m}^2\text{K}$. As there is no air flow directly impinging this region, the majority of heat is dissipated through low velocity recirculating air flows which exist in this region. In a study of the velocity field between an axial fan impinging air normal to a flat plate positioned downstream, Sui et al. (2009a) concluded that as $H/D$ was increased to 0.6, the strength of fluid recirculation beneath the fan hub also increased. Similarly, a recent experimental study of the velocity field downstream of a swirling annular jet by Yang et al. (2010) also indicated strong recirculation of fluid below the centreline region of the jet. This is also reflected in the current heat transfer results presented in figure 6.2 as strengthening flow recirculation improves the heat transfer performance to approximately $15\text{W/m}^2\text{K}$ on the flat plate directly beneath the hub centre for the largest fan to plate spacing of 15mm.

In figure 6.4 a), the standard deviation of fluctuations in heat transfer coefficient over the time-averaged data for a limited recording frequency are shown to be a maximum of 40%. A fan to plate spacing of $H/D = 0.309$ is presented using the Ø48.5mm fan, while the remaining fan to plate distances examined also produced a similar magnitude and pattern in heat transfer fluctuations. Areas under the hub experience fluctuations in heat transfer up to 80 - 100% due to the unsteadiness in air flow recirculation. The position of the motor supports at the fan exit results in an increase in heat transfer coefficient fluctuations on the flat plate, caused by the blade-support interaction. As the fan blade passes over a motor support, local pressure variations result in increased velocity fluctuations combined with wake shedding on the strut. As previously discussed, similarities between the observations at the rear of a body in cross flow and the region beneath a motor support exists. In this region, small eddies are created which are initiated as the fluid diverts the motor support. These turbulent structures continue downstream in the direction of the swirling flow until impingement on the flat surface. The maximum and minimum heat transfer fluctuations in figure 6.4 b) and c) suggest that these unsteady structures drive the heat transfer in this region. In contrast, heat transfer fluctuations in the areas unaffected by the motor support interactions have maximum fluctuations of approximately 20% with $\sigma_h$ typically less than
5\%. This implies that the effect of turbulence generated by the airfoil blades in the form of tip vortices, blade wakes and separation, has a relatively minor effect on the surface heat transfer coefficient. Of course this is presented for an acceptable fan operating point, and based on the literature reviewed (Estevadeordal et al., 2000; Velarde-Suarez et al., 2002; Grimes and Davies, 2004; Grimes et al., 2001), an increase in the magnitude of the heat transfer fluctuations will occur when operating outside the recommended design point.

The largest fluctuations are a direct consequence of the interaction of the outlet flow and the widest motor support carrying electrical wiring. These fluctuations are in a region of low heat transfer coefficient, shown in figure 6.2 c). Interestingly, the fluid interaction with the motor support provides a local increase of heat transfer coefficient $h_{fc} \approx 120 \text{W/m}^2\text{K}$, but also results in a wide band of low heat transfer coefficient $h_{fc} \approx 30 \text{W/m}^2\text{K}$ which is highly unsteady ($\sigma_h \approx 25 - 40\%$). The distribution of the maximum and minimum local
time-varying fluctuations in heat transfer coefficient is presented in figures 6.4 b) and c). The magnitude of the fluctuations reinforces the significance of unsteadiness and turbulence in the outlet flow on surface heat transfer. Often the fluctuating nature of unsteady fan flows is overlooked when analysing heat transfer performance using time-averaged information. The data presented in figure 6.4 indicates regions to avoid in the positioning of discrete heat sources, such as electronic components, with reliability which may be adversely influenced by cyclic thermal loading.

Although the primary practical application of these results is towards enhancing electronics cooling, the arrangement presented also applies to many other engineering scenarios. Akturk et al. (2009) presented an experimental study on uninhabited aerial vehicles which implement ducted axial fan designs to produce thrust to hover above ground level. In order to achieve this, the axial fan forces air normal to the surface, similar to the experimental arrangement examined in this chapter. Akturk et al. (2009) concluded that the introduction of a cross flow, representing the vehicle in forward flight, resulted in excessive moment imbalance in the axial jet. Consequently, vehicle instability is promoted with this non-uniformity in exit flow. As seen in figure 6.2 and figure 6.4, a clear moment imbalance is also evident due to the position and geometry of the axial fan motor supports, and the exit flow is asymmetric when considering the connection between fluid flow and heat transfer. The proportionality between heat and mass transfer coefficients is another analogy which extends the application of this work. In drying processes, the drying rate is dependent on fluid velocity, temperature and solvent content of the drying air (Avci and Can, 1999). Stress sensitive surfaces often require geometrically different jet designs, whose surface heat and mass transfer performance have been considered in detail in the literature by Peper et al. (1997). Figure 6.2 and figure 6.4 highlight the possible issues with axial fan impingement if drying uniformity is essential to the mass transport process.

This section illustrates a similarity in local surface heat transfer patterns for axial fans with impingement for two fan designs which are both geometrically and dimensionally dissimilar. The heat transfer trend has been attributed to the interaction of the fan outlet flow and motor supports positioned on the exit flow plane. The motor support layout for both designs are similar as shown in figure 4.2 and figure 4.3. Due to the similarities
between both designs, the following sections are focused on the miniature axial fan design of 24.6mm diameter which has dimensions suitable for space constrained and portable applications.

6.1.3 Component placement

Figure 6.2 a) - c) presents the local heat transfer coefficient over the flat surface for each fan to plate distance examined. As discussed in the previous section, the six distinctive peaks in heat transfer coefficient move with $H/D$. The importance of component placement is stressed when examining the mean heat transfer performance of a representative component in a portable device with dimensions $5\text{mm} \times 5\text{mm}$, positioned centrally on these peaks in heat transfer. This is shown diagrammatically in figure 6.5 for a $H/D = 0.203$ example. An additional component position is also included in figure 6.5, labeled 0, which is located directly on the fan central axis. Components are orientated parallel with the fan housing, which is a realistic situation in board level cooling.

![Diagram of component locations](image)

Figure 6.5: Component locations 0 to 6 for $H/D = 0.203$ and 9000 rpm.

The mean heat transfer coefficient for each component is presented in figure 6.6 for $0.203 \leq H/D \leq 0.610$. Positioning the component on peak 6 provides the greatest enhancement in heat transfer for the entire $H/D$ range. This optimum component location is a direct consequence of the air flow interaction with the largest motor support, which carries
the electrical wiring to the motor. The component located on peak 5 is centred on the secondary peak in heat transfer coefficient due to the air flow and motor support interaction of the largest motor support. The lowest heat transfer performance is achieved by positioning a 5mm × 5mm heat source at this location over the alternative locations 1 – 4, and 6. At \( H/D = 0.203 \), a 25% increase in mean heat transfer coefficient is possible by positioning the component at location 6 over location 5. However, this level of enhancement decreases with increasing \( H/D \), and at \( H/D = 0.610 \), an increase of 12% is apparent. Interestingly, at \( H/D = 0.610 \), locations 1 – 5 provide almost identical heat transfer performance. For all \( H/D \) investigated, positioning the component at location 0 provides the least heat transfer performance at 35W/m²K. However, just 12mm away, a possible 125W/m²K can be achieved at location 6.

In combined axial fan and heat sink cooling solutions, the heat source is generally located at a central position on the heat sink base. This typically corresponds to the central axis of the fan, as the footprint area of the heat sink normally resembles similar dimensions as the fan housing (Lin et al., 2005). As a result, the heat source is positioned at location 0, and sufficient heat removal is therefore dependent on the effective thermal spreading properties of the heat sink base to maintain component reliability. In addition to this, if the footprint area of a finned design heat sink is similar to the fan housing area, the peaks in heat transfer, shown in figure 6.2, will occur outside the heat sink base for all \( H/D \) examined.
This highlights possible improvements which can be introduced to existing designs for further enhancement in heat transfer performance. In general, component locations 2, 4, and 6 appear to be the preferred positions for component placement. The higher heat dissipation rates in these regions are due to the physical deflection of the exit air flow by the motor supports. In figure 6.3, the possible deflection of air flow as a fan blade rotates past a motor support is described. The air flow from the pressure side of the fan blade is forced to accelerate around the motor support, resulting in high velocity regions that move downstream, impinging the flat surface, and producing regions of increased heat transfer. The approach angle of these two regions of increased air flow towards the plate however, may be different. As the blade approaches the near side (NS) of the motor support, the swirl angle may be decreased if the motor support thickness is large enough to cause a deflection. As the blade passes over the motor support and approaches the far side (FS), the fluid retains its original swirl angle. Consequently, the air which impinges at component locations 2, 4, and 6 travels a shorter distance when impinging the flat surface, over the air which impinges at 1, 3, and 5. The axial velocity is therefore higher at 2, 4, and 6, producing higher heat transfer coefficients than at 1, 3, and 5. This difference in swirl angle is confirmed in section 6.1.4 which examines the exit flow characteristics of the axial fan.

6.1.4 Exit flow characteristics

Swirl and jet angle measurements can be used to determine the exit flow characteristics of the axial fan, and compared with the heat transfer phenomena on the heated flat surface. In order to determine the swirl angle of the exit flow, a tuft of nylon thread ≈0.5mm wide, and 7mm in length, was introduced into the air flow. The tuft was allowed to move freely and therefore aligned itself with the fan outlet flow. The angle between the absolute velocity direction (tuft direction) and the axial velocity direction was then measured. The swirl angle was measured on the exit flow plane between the two smaller motor supports (figure 4.2), and 60 degrees from either support. At this measurement point, the adverse influence of the fluid and motor support interaction, on the swirl angle, is at a minimum.

The axial jet angle was investigated using smoke visualisation. A Rosco 1700 fog machine was used to introduce smoke into a compartment that supplied air to the axial fan.
The axial fan extracted the smoke and air mixture from this compartment and exited to the ambient air. Therefore on exit from the fan, it was possible to distinguish the conical flow profile of the exit flow, and measure the jet angle.

Grimes et al. (2001) examined the influence of static pressure rise on jet angle of an axial flow fan for system impedance levels of approximately 20%, 40%, and 60% of the maximum fan static pressure. It was shown that at the lower operating point (20%), the radial angle of the flow at the fan outlet was almost unaffected by the increase in static pressure, and the growth of the jet was mainly due to the growth of the free turbulent mixing layer. As the static pressure for the entire range examined only reaches approximately 3% of the maximum static pressure attainable by the axial fan (figure 6.1 a)), it was assumed that variations in the exit flow characteristics due to this static pressure rise were minor and within the uncertainty of the measurement technique. The swirl and jet angle measurements were therefore considered for the simplified arrangement of an axial fan operating in standalone with zero system impedance.

An image of the axial jet, produced for a rotational speed of 9000 rpm, is presented in figure 6.7. The measured jet angle, $\beta$, is also indicated. In table 6.1, the jet angle is provided from the smoke visualisation tests for all fan speeds examined. The angle varies from $21^\circ$ to $22^\circ$, which indicates the cooling area of the axial jet is almost constant at a particular $H/D$ for all fan speeds. This is confirmed through analysis of the cooled area from the radial average heat transfer distributions over $0.203 \leq H/D \leq 0.610$. Similar jet angles as shown for the smoke visualisation investigations, between $21^\circ$ and $23^\circ$, have also been noted in

![Figure 6.7: Axial jet produced at a fan rotational speed of 9000 rpm.](image-url)
Table 6.1: Jet and swirl angles of exit flow.

<table>
<thead>
<tr>
<th>Fan speed (rpm)</th>
<th>$\beta$ (smoke vis.)</th>
<th>$\beta$ (heat transfer)</th>
<th>$\alpha$ (tuft meas.) (regions 1, 3, 5)</th>
<th>$\alpha$ (regions 2, 4, 6)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3000</td>
<td>21</td>
<td>21</td>
<td>58</td>
<td>-</td>
</tr>
<tr>
<td>6000</td>
<td>22</td>
<td>23</td>
<td>42</td>
<td>38</td>
</tr>
<tr>
<td>9000</td>
<td>21</td>
<td>21</td>
<td>40</td>
<td>37</td>
</tr>
<tr>
<td>12000</td>
<td>22</td>
<td>21</td>
<td>39</td>
<td>37</td>
</tr>
</tbody>
</table>

The radial distributions highlight a heat transfer maximum at a radial location on the plate corresponding to the location where the maximum local velocity of the jet, near the jet edge, impinges the surface. The radial location of this heat transfer maximum increases with increasing $H/D$, due to the jet angle, $\beta$, and is discussed further in the proceeding section. The radial position of the maximum heat transfer with changing $H/D$ can therefore be estimated knowing the jet angle. The smoke visualisation method for determining the axial jet angle is therefore a simple, yet useful method to establish the cooling area of an axial fan.

The local heat transfer data have also been used to estimate the swirl angle of the axial fan jet, and confirm the discussion on the variance in the magnitude of the 6 peaks in heat transfer in section 6.1.3. Figure 6.8 presents helical expanding, exit flow streamlines near the outer edge of the jet created by a rotating axial fan. One of these streamlines has been highlighted to discuss the calculation of swirl angle from the heat transfer data. This streamline represents the path of maximum local velocity downstream of the fan outlet. Three planes normal to the jet, and corresponding to the range of $H$ examined, are outlined. The intersection of these planes with the streamline indicates the point of impingement on the flat plate and where a local maximum in heat transfer occurs. These intersection points are indicated for $H = 5 – 15\text{mm}$, and also included in a plan view above the fan outlet. It is possible to infer an estimation of the swirl angle from these points of local maximum heat transfer, knowing the change in circumferential position of an intersection point due to a change in plate distance from the fan, $H$. 


In figure 6.9, the movement of the peaks in heat transfer coefficient with increasing $H/D$ is presented for fan speeds 3000 - 12,000 rpm. The six regions identified, correspond to the positional changes of the six peaks in heat transfer with $H/D$, previously discussed with the aid of figure 6.2 a) - c). In each region, the peaks move away from the fan outlet in the radial direction, while also in the fan rotational direction. These helical traces collapse on top of one another over the entire $H/D$ for fan speeds 6000 – 12,000 rpm, however the remaining trace for 3000 rpm is much more scattered. This fan speed corresponds to $Re_b = 2000$, the Reynolds number found by others to change the aerodynamic scaling characteristics (Neustein, 1964), which results in an increase in the swirl angle of the exit flow. The tuft measurements, presented in table 6.1, confirm this, as a significant increase in $\alpha$ is experienced at 3000 rpm. The swirl angle for the remaining fan speeds, which correspond to $4000 < Re_b < 8000$, are similar. This is reflected by the collapse of the peaks in heat transfer, shown in figure 6.9.

A distinctive difference between the regions defined 1, 3, and 5, and regions 2, 4, and
Figure 6.9: Helical trace of peaks in heat transfer coefficient with increasing $H/D$ for all fan speeds examined.

6 exists. The angles the peaks in heat transfer change position with increasing $H/D$ differs between these two groups. The swirl angle of the exit flow can be estimated from the heat transfer measurements by tracking the movement of these peaks over the various $H/D$, shown in figure 6.9. The averaged results for the swirl angle in regions 1, 3, and 5, and regions 2, 4, and 6, are shown in table 6.1. Measurements for a 3000 rpm fan speed are not considered, due to the scatter in the heat transfer peak positions, although it confirms the low $Re_b$ effect. The estimation for regions 1, 3, and 5, are within 4° of the tuft measurements, whereas the swirl angle in regions 2, 4, and 6, are up to 10° less. The exit flow interaction with the motor supports is therefore influencing the approach angle of the impinging air flow, discussed in the previous section on component positioning, with the aid of figure 6.5. The near side of the motor support deflects the air flow, reducing the swirl angle, and consequently producing higher magnitude heat transfer peaks than the remaining regions 1, 3 and 5.

### 6.1.5 Radial heat transfer

The mean non-dimensional radial heat transfer distribution is presented in figure 6.10 for the nominal fan speed and $H/D$ range examined. This expresses the average trend in heat transfer over a radius to fan diameter ratio $r/D$, from 0 to 1.6. Fan hub and blade locations are also indicated. Beneath the hub region, the Nusselt number is at a minimum on the fan
central axis and increases with increasing $r/D$. As $H/D$ increases, the maximum Nusselt number decreases, and shifts in the direction of increasing $r/D$. This is due to the diverging angle of the jet, shown in figure 6.7, which also decelerates downstream of the fan outlet. This diverging air flow angle corresponds with the angle the peak locations in radial heat transfer produce with increasing $H/D$. The variation of $Nu_d$ with fan rotational speed is shown in figure 6.11 for $H/D = 0.203$. The maximum radial heat transfer performance occurs at $r/D = 0.5$ for all speeds, which corresponds to the outer edge of the fan jet at this distance from the surface. This peak in heat transfer occurs as the momentum of the axial fan outlet flow is at a maximum near the outer edge of the fan jet as a result of the centrifugal forces within the jet. At $r/D > 1$, $Nu_d$ begins to approach a constant value for the 3000 rpm case, and cooling enhancements provided by the impinging jet are no longer noticeable.

The distribution of the Nusselt number with radial position for annular jets has been shown to scale with $Re^{0.55}$ (Chattopadhyay, 2004). An analogy between axial fan Reynolds number ($Re_d$) and resultant surface heat transfer is presented in figure 6.12 for both fans considered in this chapter. The scaling relationship is found to be $Re^{0.6}$ for impinging axial fan flows as shown in figure 6.12 and applies to the entire $H/D$ range examined. The increase in the Reynolds number exponent over the annular jet study by Chattopadhyay (2004) reflects the unsteady nature of axial fan flows over similar Reynolds number annular jet flows. This is confirmed by the previous discussion on the heat transfer fluctuations.
which exist on the surface of the flat plate. The primary source of this unsteadiness has been accredited to the interaction between the outlet flow and motor supports on the exit plane. The influence of low velocity fluctuations beneath the fan hub are also highlighted in figure 6.12, as the heat transfer data deviates from this relationship with Reynolds number. As previously discussed, a complex diverging, swirling, and annular jet is produced by an axial fan. To understand the significance of the scaling relationship between Nusselt number and Reynolds number in figure 6.12, it is necessary to discuss the features of the free jet before it impinges the flat surface. Both the inner and outer edges of the axial jet
mix with the surrounding ambient air on approach to the impingement surface. This mixing effect at the jet edges results in the growth of a turbulent shear layer downstream of the fan outlet. Within this shear layer, turbulent fluctuations are at a maximum for the free axial jet. This local increase in turbulence is reflected in the heat transfer measurements at the locations where the inner and outer jet shear layers interact with the flat plate. The scaling relationship produced by the core axial jet flow, where heat transfer is a maximum, is found to be $\text{Nu}_d \sim \text{Re}_d^{0.6}$ for both Ø24.6mm and Ø48.5mm axial fans in figure 6.12 a) and b). At $r/D < 0.2$ and $0.8 < r/D < 1.6$, the scaling relationship deviates from $\text{Re}_d^{0.6}$ to $\text{Nu}_d \sim \text{Re}_d^{0.8}$, as the jet inner and outer shear layers impinge the surface in these regions. For $0.8 < r/D < 1.6$, this relationship is maintained to the limiting $r/D$ examined. Therefore, although the aerodynamic properties of the flow are sensitive to Reynolds number, it appears that similar scaling can be used with the range of $60 < \text{Re}_d < 3000$ to predict the radial heat transfer performance of axial fans.

### 6.2 Motor support effects

Section 6.1 presented the local heat transfer coefficient for two different axial fan designs with a similar motor support layout at the outlet plane of the fan (figures 4.2 and 4.3). The influence of the motor supports on the non-uniformity in surface heat transfer has been discussed in detail. As a significant contribution is evident, the influence of motor support positioning on heat transfer is an important parameter, in particular at the miniature scale, as motor support dimensions are independent of fan diameter. Therefore, motor supports for miniature axial fans may occupy a greater portion of the annular fan outlet area than at larger scales.

As there is a necessity to support the motor at the centre of the hub, there are typically two arrangements; Motor supports positioned on the outlet flow plane or motor supports positioned on the inlet flow plane. The effect of motor support arrangement on acoustic emissions has received some attention in the literature, where the main contribution is due to the fan blade interaction with the thicker motor support carrying the electric cables from the motor to the housing (Lu et al., 2007; Huang and Wang, 2005). The influence
motor support positioning has on heat transfer performance at miniature scales has been considered by comparing two additional arrangements to the outlet positioned motor support case examined in section 6.1.2. A modification of the radial fan experimental rig was used to rotate the axial fan within a housing which had motor supports positioned on the inlet plane and also for a second arrangement which had no supports on the housing. This experimental assembly is shown in the appendices in section B.3.

In figure 6.13, the heat transfer coefficient on a flat surface is presented for each motor support configuration examined. Comparing the data for outlet and inlet mounted motor support configurations in figure 6.13 a) and b), six peaks in heat transfer coefficient are produced on the surface due to the fan blade interaction with the motor supports. The most noticeable difference however is a more uniform surface heat transfer distribution

![Figure 6.13: Forced convection heat transfer coefficient for a) motor supports on outlet plane, b) motor supports on inlet plane, and c) no motor supports at 9000 rpm and $H = \text{5mm}$. d) Mean radial distribution of heat transfer coefficient for a) - c).](image-url)
for the inlet mounted motor support arrangement. The offset of the peaks in heat transfer coefficient shown in figure 6.13 a) has been discussed in section 6.1.4 and attributed to the swirling exit flow of the axial fan. In contrast, the peaks produced due to inlet mounted motor supports are no longer offset in the direction of fan rotation. This indicates the inlet air flow approaches the suction side of the fan blades in a predominantly axial direction, without a swirl component, which is only introduced by the pressure side of the fan blade, deflecting the air flow downstream. Similarly for both arrangements, the largest motor support, carrying the electric wiring, deteriorates the heat transfer coefficient the most as it is almost 50% of the width of each passing fan blade.

Figure 6.13 c) presents the local heat transfer coefficient for the miniature axial fan design without any motor supports. The result is an annular region of increased heat transfer, which is circumferentially uniform in all radial locations from the fan centre. In addition, there is a substantial increase of approximately 20% in average radial heat transfer coefficient by excluding motor supports from the fan outlet and inlet flow planes. This is presented in figure 6.13 d) for all cases a) - c). A relatively minor increase in heat transfer coefficient is observed by using a design which has motor supports on the inlet flow plane over a design which uses supports on the outlet. Better uniformity of the local heat transfer distribution in the circumferential direction can also be preferable as seen in figure 6.13 b). Therefore, for miniature axial fans, designs should be considered which include motor supports positioned on the inlet flow plane. These motor supports should also be of slim dimensions without compromising mechanical strength, thereby lessening the blockage effects, and increasing the heat transfer performance and uniformity of the outlet air flow.

6.3 Closure

This chapter examined the local surface heat transfer distributions for axial fans impinging air normal to a flat plate. The interaction of air flow leaving the fan blades with the motor supports on the exit flow plane of an axial fan were shown to result in localised maxima in the heat transfer coefficient. In addition, fluctuations in surface heat transfer coefficient
were observed, and directly linked to the unsteady fluid structures generated by this interaction. Subsequently, significant enhancements in heat transfer can be achieved with accurate heat source positioning. Optimum placement of components was investigated, and enhancements in the mean heat transfer coefficient were observed through relatively small positional changes.

An increase in fan to plate distance results in local peaks in heat transfer moving outwards on the plate surface from the fan central axis. This has been attributed to the conical exit flow distribution. The locations of these regions of high heat transfer were independent of fan rotational speed above a critical Reynolds number, where the aerodynamic performance of the fan is sensitive to the ratio of viscous to momentum forces. The exit flow characteristics of an impinging miniature axial fan jet were examined to understand the heat transfer phenomena on the flat surface. For the two different axial fan designs presented, the radial distribution of non-dimensional heat transfer was shown along with a common scaling relationship with Reynolds number.

With motor supports positioned on the exit flow plane resulting in substantial non-uniformity of the local heat transfer coefficient, an investigation was conducted to determine the preferred arrangement for a miniature axial fan. An additional inlet mounted motor support arrangement was considered along with an ideal case without any motor supports present. It was concluded that for miniature fan designs, the inclusion of motor supports can reduce thermal performance significantly and up to 20% reductions in the radial distribution in heat transfer coefficient were noted for the example presented. Based on the findings it was concluded that motor supports should be positioned on the inlet flow plane of miniature axial fan designs, and also have minimal dimensions for structural stability to avoid substantial blockage effects.
Chapter 7

Heat transfer with Radial fans

Bulk heat transfer measurements presented in Chapter 5, using radial fans enclosed within a housing, indicated that finless heat sink designs can be a beneficial design choice for low profile, space constrained environments. In the investigation of component placement it was also highlighted that a radial fan without the confinement of a housing could provide high levels of heat dissipation in a radial finless heat sink design and conserve the overall foot print area of the cooling solution. This type of cooling solution arrangement is considered in greater detail in this chapter by relating local velocity field and heat transfer measurements for similar forward curved fan designs with various profiles and diameters.

Velocity field measurements are presented for geometrically similar fans with diameters ranging from 15mm to 59mm using the experimental PIV apparatus and technique outlined in section 4.3. The influence of varying fan aspect ratio and flow rate on the unsteady flow field within the finless channel are assessed to determine the fluidic mechanisms which govern the heat transfer distribution within the finless design. Local heat transfer distributions of the base and top plates of a radial finless heat sink design have been obtained using combined infrared thermography and a heated-thin-foil measurement technique presented in section 4.1.4. The relationship between fan performance characteristics and local heat transfer performance is also examined to determine the limits of fan profile scaling on thermal performance. In addition, scaling of fan diameter for increased flow rate has been considered to extend the use of the low profile finless cooling solution from portable devices to larger scale applications with space constraints.
Chapter 6 detailed the thermal performance of a miniature axial fan with impingement normal to a heated surface at low profile spacings suitable for portable applications. This chapter contains direct comparisons between the heat transfer performance using axial fan jet impingement and heat transfer with miniature radial fans to conclude on a preferred design choice for portable applications based on power usage and cooling solution space requirement as the main criteria.

The final part of this chapter investigates the influence of practical operating conditions, such as crossing air flows and fan inlet confinement, on the local and area average heat transfer performance of a finless cooling solution. A uniform cross flow was introduced above the fan inlet for two designs with diameters 15mm and 59mm. Velocity field and local heat transfer measurements have been implemented to assess the effect of cross flow magnitudes from 0 to approximately 2m/s and fan inlet confinement cases which are detailed in table 4.6. Local heat transfer distributions have been averaged over an area representing an electronic component located at various positions around the fan to evaluate the recommended component placement for maximum thermal performance when the cooling solution is subjected to a crossing air flow. Finally, circumferential and area averages of the local heat transfer distributions are presented to highlight the degrading effect of a cross flow on the overall cooling solution thermal performance.

7.1 Radial fans and finless heat sink

The objectives in this section are to firstly examine fan performance and highlight the scaling phenomena that exist at certain fan blade Reynolds numbers and adversely affects fan performance characteristics. The velocity field within a radial finless heat sink is then presented for a range of diameter scaled fans. Finally, local and area average heat transfer distributions are examined to determine the influence of combined fan and heat sink profile scaling, fan diameter scaling, and fan inlet confinement on thermal performance attributes. The distance between the plates of the finless channel is $H_f + 1\text{mm}$, to provide clearance between the fan and base plate (0.5mm), and the fan and top plate (0.5mm). Comparisons between the heat transfer performance using axial and radial fan designs are also provided.
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

7.1.1 Pressure and flow rate characteristics

In section 4.1.4, the geometric details are provided (table 4.3) on the radial fans considered in the analyses of local heat transfer distributions within a radial finless heat sink. Six different fans were examined which have aspect ratios, $0.068 \leq a_r \leq 0.433$. The non-dimensional flow and pressure coefficients have been defined in Eq. (3.8-3.9), and are plotted in figure 7.1 for all fan designs of table 4.3 operating at blade Reynolds numbers $Re_c > 1000$.

Figure 7.1 indicates the issue associated with accurately predicting fan characteristics for a range of aspect ratios using conventional scaling laws. This observation was noted by Walsh et al. (2009b) who presented results for a constant diameter forward curved radial fan with various aspect ratios from 0.01 to 0.63. As aspect ratio was decreased, the pressure coefficient was shown to decrease which is also shown in figure 7.1. Walsh et al. (2009b) determined that the fan scaling laws could only predict the performance of the fan design investigated for the limited range $0.12 \leq a_r \leq 0.17$. The Ø15mm fan with $H_f = 2\text{mm}$, and Ø24mm fan with $H_f = 4\text{mm}$ are within this range and appear to resemble similar non-dimensional performance attributes with maximum flow and pressure coefficients within 14%.

Figure 7.1 also highlights the importance of selecting a blade profile within the maximum limit where the inlet chokes the flow and no further benefits in flow rate are experienced. The maximum blade profile recommended by Bleier (1997) for forward curved blade designs is $H_{f-max} = 0.6D_{in}$ ($a_r = 0.431$) which equates to 6.46mm for the Ø15mm fan design. In contrast, Walsh et al. (2009b) found that above an aspect ratio of 0.35 no

![Figure 7.1: Non-dimensional pressure and flow coefficient for all radial fan diameters.](image-url)
benefit in flow rate was achieved. This is confirmed in the maximum flow rate measurements of the Ø15mm fan with $H_f = 6.5\text{mm}$ which provides minimal increase in flow rate over the Ø15mm fan with $H_f = 4\text{mm}$. These measurements are presented in figure 7.2 for a range of rotational speeds. Consequently, velocity field measurements on the Ø15mm fan were only considered for the maximum flow rate fan design with $H_f = 4\text{mm}$. Velocity field measurements were also considered for the other fan diameters with $H_f = 4\text{mm}$ to examine the influence of decreasing aspect ratio on the flow field.

Although a profile height of 2mm reduces the flow rate, the static pressure level is maintained as the flow accelerates during the $90^\circ$ turn in the fluid from an axial to radial direction. An acceleration in flow occurs when the inlet area to the blade passages ($\pi D_m H_f$) is less than the inlet orifice area ($\pi D_m^2 / 4$). This is reflected both in figure 7.1, and also figure 7.2 for a range of fan speeds. The linear regime in figure 7.2 a) and power law trend in figure 7.2 b) also confirms the scaling relationship with fan rotational speed presented in Eq. (3.1-3.2).

Scaling effects with reduced blade Reynolds number, have been shown to influence the local heat transfer distributions in the previous chapter on axial fans. Similarly, the miniature radial fan designs investigated in this chapter are also subjected to the same phenomenon. In figure 7.3, the relationship between Reynolds number and the non-dimensional coefficients of flow and pressure are described. The flow coefficient reduces significantly at low Reynolds numbers, and appears to be reaching a constant value for $Re_c > 1000$. The

Figure 7.2: The trend in maximum a) flow rate and b) static pressure for a range of fan speeds and fan profiles with Ø15mm.
pressure coefficient holds almost constant with slight increases at the lower Reynolds numbers. This scaling effect on aerodynamic performance was noted by Walsh et al. (2010) for similar fan designs as examined here and was attributed to a reduction in mean velocity in the blade channels due to the boundary layers which exist on the pressure and suction sides of the blade.

Bulk flow rate and pressure characteristics of radial fans used to investigate thermal performance of a radial finless heat sink geometry in this chapter have been presented. In the following section, the local velocity field of the fan outlet flow is presented for the range of diameters 15 - 59mm with a constant profile height of 4mm. This outlet flow results in forced convection along the surfaces of the finless top and base plates. With the aid of velocity field measurements, the fluidic mechanisms that enhance forced convection can be visualised.

### 7.1.2 Velocity field

This section investigates the velocity field generated by fans of diameter 15mm, 24mm, 32mm, and 59mm for a constant fan profile height of $H_f = 4$mm. These fans were integrated into a finless heat sink geometry consisting of parallel base and top plates with a spacing of 5mm to allow for fan clearance. This arrangement has been shown previously in the experimental apparatus of figure 4.29 which illustrates the experimental layout for the
measurement of radial \((u)\) and axial \((w)\) components of the velocity field within the finless channel for an axisymmetric fan exit flow. For each fan examined, the velocity fields are presented for a radial distance from the fan blade tip, \(r - 0.5D\), from 0 to 0.04m.

Figure 7.4 presents the time-averaged velocity magnitude and streamlines in the radial-axial plane between base and top plates when using a Ø15mm fan. The air flow exits the fan blades producing a high shear flow with a large velocity gradient on the base plate, indicated by the normalised velocity profile of figure 7.4 d) at a location 0.005m from the fan blades. All of the velocity profiles presented in this section have been normalised with the maximum velocity magnitude at \(r - 0.5D = 0.005m\). The high shear flow enters the finless channel at approximately one half of the channel height, and extends in the radial direction to a radial distance which is dependent on fan speed. This type of fan exit flow profile results from much of the airflow tending towards the back plate of the fan due to the inertia forces when the fan redirects the axial inlet flow 90° to produce a radial outlet flow. For the range of speeds examined using this fan, a primary vortex is evident which is promoted by the high shear layer interacting with the lower velocity fluid in the remaining half of the channel. In addition, the deflection of fluid, which impinges on the top plate, back towards the fan blades also contributes to this time-averaged vortical motion. The existence of a counter rotating secondary vortex downstream results in an adverse pressure gradient in the radial direction, as the secondary vortex opposes this high velocity flow along the surface of the base plate. Consequently, the high shear flow separates from the base plate as the secondary vortex reduces the velocity in the boundary layer. This reduction is also observed in figure 7.4 d) when examining the velocity profile at 0.015m from the fan blades at 10,000 rpm. Between 0 and 0.0015m from the base plate, a small peak in velocity magnitude is shown which is the velocity of the reversed flow of the secondary vortex. Above this point in the channel, the boundary layer of the high shear flow is evident. The existence of both time-averaged vortices promotes fluid impingement on the top plate which has been briefly discussed in section 5.3 of Chapter 5. It is apparent when comparing figure 7.4 a) - c) that increasing fan speed, and therefore flow rate, shifts the point of impingement in the radial direction away from the fan. This shift is due to an increase in pressure in the fan exit flow over the adverse pressure caused by the secondary vortex.
Figure 7.4: Time-averaged velocity magnitude and streamlines using a Ø15mm fan with \( H_f = 4 \text{mm} \) (\( \alpha_r = 0.267 \)) operating at a) 1000 rpm, b) 5000 rpm, and c) 10,000 rpm. d) Normalised velocity profiles of c) for a series of radial locations from the fan blades.
Consequently, at 10,000 rpm, the secondary vortex appears to be getting squeezed against the base plate in the axial direction. The impingement on the top plate deflects the air flow towards the base plate which is aided by the momentum of the secondary vortex, resulting in a second impingement on the base plate.

The time-averaged turbulence statistics are presented in figure 7.5 in addition to instantaneous velocity magnitude and streamline distributions at 0 and +1 seconds for 10,000 rpm. A comparison between the instantaneous flow fields of figure 7.5 a) and b) indicates the high level of unsteadiness which exists in the finless channel. The instantaneous distributions also highlight the contribution of the counter rotating vortex on the radial pressure gradient. In figure 7.5 a) this vortex is not present in the flow and the high shear flow develops in the radial direction without impingement on the top plate. In figure 7.5 b) however, the existence of this vortex causes an impingement on the top plate, as the high velocity fluid is forced from a radial to predominately axial direction.

Vortices which are evident in the upper half of the finless channel also appear to be intermittently disrupting the high velocity shearing flow along the base plate. This is caused by axial movement of vortices which encompass a level of vorticity that results in acceleration and deceleration of the shear flow upon interaction. This is observed in the instantaneous velocity magnitudes in figure 7.5 as regions of increased and decreased velocity magnitude are apparent from 0 to 0.02m from the fan blades. The radial and axial turbulence intensities, $I_r$ and $I_z$, increase up to 60% of the mean fan inlet velocity as a result of this interaction. Furthermore, the magnitude of Reynolds shear stress in the fan blade exit flow region, shown in figure 7.5 e) also confirms the high velocity shear flow is affected by the interaction of the unsteady vortical structures. The magnitude of shear stress is maximum along the shear layer formed between the high velocity exit flow and the lower velocity vortical structures in the upper half of the finless channel. Finally, at extended radial locations $(r - 0.5D) > 0.025m$, vortices begin to breakdown with dissipation of rotational momentum into the mean flow. The mean radial flow in the finless channel appears to approach a parabolic profile as shown in figure 7.4 d).

The Ø24mm fan produces a velocity field similar to that of the Ø15mm fan and has been presented previously in Chapter 5 for two rotational speeds of 4300 rpm and 7500
Figure 7.5: Velocity field statistics within a finless heat sink design using a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm. a), b) Instantaneous velocity magnitudes at 0s and +1s; c) Turbulence intensity in the radial direction; d) Turbulence intensity in the axial direction; and e) Reynolds shear stress in the radial-axial plane.
rpm (figure 5.14). The significant difference however, is the absence of the secondary vortex within the radial-axial flow field. By operating the Ø24mm fan at rotational speeds above approximately 4000 rpm, it was found that the secondary vortex was absent from the flow field due to the effects of increased flow rate between the finless channel previously discussed in this section. Although separation from the base plate, produced by the existence of the secondary vortex, is not apparent above 4000 rpm, the velocity profile appears to be directed towards the top plate to some extent between 0.01m and 0.015m from the fan blades. It is anticipated that the pressure variation across the curved streamlines of the primary vortex results in a net force acting perpendicular to these streamlines and towards the centre of curvature (Massey, 2006). This net force causes the high shear flow to bend towards the top plate, following the curvature of the primary vortex rather than continuing in a solely radial direction along the base plate.

An example of the velocity field within the finless channel using a Ø32mm fan operating at 1000 rpm is presented in figure 7.6. Again, this shows similar fluidic mechanisms exist with fan diameter scaling. At fan speeds above approximately 1700 rpm, the secondary vortex no longer exists in the time-averaged velocity field. As fan diameter is scaled further

![Figure 7.6: Time-averaged a) velocity magnitude and streamlines and b) normalised velocity profiles using a Ø32mm fan with $H_f = 4\text{mm}$ ($a_r = 0.125$) operating at 1000 rpm.](image)
for this constant fan profile height, the time-averaged vortices become less noticeable as large increases in flow rate are achieved at lower fan rotational speeds \( Q \propto \omega D^3 \). For the Ø59mm fan therefore, the secondary time-averaged vortex only exists in the finless channel at fan rotational speeds below 750 rpm. When the secondary vortex is absent from the flow field, the local region of impingement on the base plate is removed and may result in reduced thermal performance at this local position. However, by overcoming the opposing secondary vortex the detachment of the high velocity flow exiting the fan no longer occurs, and velocity gradients at the surface are increased which would lead to heat transfer enhancement. In the following section, the influence of this secondary vortex on local heat transfer within the finless channel is discussed. The time-averaged velocity magnitude and streamlines above this rotational speed are represented in figure 7.7 for a fan rotational speed of 1000 rpm. The location of a single vortex is shown to accomodate a much smaller area of the flow field than the primary vortex for the larger aspect ratio fans. The normalised velocity profiles of figure 7.4 b), figure 7.6 b) and figure 7.7 b) suggest that as fan aspect ratio is decreased for a constant plate spacing, the flow within the channel approaches a near parabolic profile much sooner than for aspect ratios

![Figure 7.7](image-url)

*Figure 7.7: Time-averaged a) velocity magnitude and streamlines and b) normalised velocity profiles using a Ø59mm fan with \( H_f = 4\text{mm} \ (a_r = 0.068) \) operating at 1000 rpm.*
$a_r > 0.125$. This is particularly evident when using the Ø59mm fan with $a_r = 0.068$. A theoretical velocity profile for a distance of 0.035m from the blade tip is included in figure 7.7 based on the measured flow rate and the Hagen-Poiseuille profile for fully developed flow between parallel plates. At distances greater than 0.015m from the fan blades, the experimental velocity profile almost resembles a parabolic shape with a maximum velocity near the centre of the channel. In contrast to the larger aspect ratio fans, the outlet flow for the Ø59mm fan is more uniformly distributed over the entire blade height.

The instantaneous velocity fields of figure 7.8 a) and b) highlight the unsteadiness associated with the outlet flow. Vortices occur predominately in the upper half of the finless channel, as observed for the smaller fan diameter with larger aspect ratio. Similarly, these unsteady fluid structures disrupt the shearing outlet flow. The occurrence of high and low velocity magnitude regions within the finless channel can be associated with this interaction, and also with the pressure fluctuations created by the passing of fan blades circumferentially. This results in a pulsed-type flow with a frequency depending on fan rotational speed for all fan diameters examined having a constant number of blades (18). The highest levels of turbulence intensity and Reynolds shear stress from these contributions in the fluid flow were noted up to 0.015m from the fan blade tip as shown in figure 7.8 c) - e).

For all fan diameters investigated, the velocity field analyses have illustrated an unsteady flow within the finless channel which contains vortex structures that disrupt the high velocity shearing flow in the entrance region to the channel. This disruption of the boundary layer along the plate surfaces can enhance heat transfer with turbulent diffusion. In the following section, the local and radial heat transfer distributions are presented for each fan diameter to determine the level of enhancement the unsteady flow within the channel produces.
Figure 7.8: Velocity field statistics within a finless heat sink design using a Ø59mm fan with $H_f = 4$mm operating at 1000 rpm. a), b) Instantaneous velocity magnitudes at 0s and +1s; c) Turbulence intensity in the radial direction; d) Turbulence intensity in the axial direction; and e) Reynolds shear stress.
7.1.3 Local and average surface heat transfer

In section 7.1.1, the variation in fan performance characteristics with fan profile and diameter scaling was briefly discussed. A detailed discussion of the flow fields generated by fans of various aspect ratios within a finless heat sink design was presented in the previous section. This section examines both the influence of fan profile and diameter scaling on the thermal performance of a finless cooling solution through measurements of local and radial distributions in heat transfer coefficient. The apparatus for the measurement of local heat transfer coefficients using a radial fan and finless heat sink has been presented in section 4.1.4. This was used to determine if a wide range of fan aspect ratios, with the same geometric design, provided similar heat transfer trends on the surface that could be related to the velocity fields illustrated in the previous section.

In Chapter 6, air impingement normal to a flat surface using a miniature axial fan was shown to provide local heat transfer coefficients of up to $140\text{W/m}^2\text{K}$ at low profiles suitable for space constrained environments. A comparison between the thermal performance of the miniature axial fan design and the miniature radial fan designs considered in this chapter is presented in this section also.

Space constrained electronic applications often result in the cooling solution being located within a limited volume that can be confined by nearby covers or circuit boards. This can result in a blockage effect on the fan inlet flow, reducing the mass flow through the fan and heat sink, and deteriorating the heat transfer performance of the cooling solution. The influence on local heat transfer of a plate positioned above the fan inlet to confine the inlet flow is considered in the final part of this section.

Profile scaling of fan and finless design

Although fan profile height has been shown to be an important parameter in the selection of fan designs, the influence of this parameter on heat transfer augmentation in a finless heat sink design has not yet been confirmed. In section 7.1.1, varying fan profile alone for a miniature Ø15mm fan resulted in a wide range of fan performance attributes. In figure 7.9 and figure 7.10, the base and top plate heat transfer coefficient distributions are presented for the Ø15mm fan at 10,000 rpm with profile heights $H_f$ of 2mm, 4mm and
6.5mm respectively. Also included are the corresponding distribution of fluctuations in the surface heat transfer coefficient, $\sigma_h$. The outline of the fan has been superimposed on all local distributions for the purpose of discussion.

The magnitude of the base plate heat transfer coefficient shows that similar heat dissipation levels are achieved in the shearing flow region for all fan profiles. In figure 7.9 a) and b) for the 2mm and 4mm fan profiles, the area occupied by the secondary vortex is apparent, as well as an annular region of increased heat transfer coefficient produced by the secondary impingement, as observed in the velocity field measurements of figure 7.4. Using the 4mm profile fan, a greater surface area is covered by the shearing flow and secondary impingement zones due to the increased flow rate for this fan profile. At this fan rotational speed, the 6.5mm fan profile has a similar heat transfer distribution however with the absence of an increase in $h_{fc}$ due to a secondary impingement. It is anticipated that the secondary vortex does exist in the flow field, resulting in a detachment as observed for the other profiles. However, due to the larger channel spacing combined with no increase in flow rate over the 4mm profile fan, the initial impingement on the top plate is at a much lower velocity. This is confirmed when examining the top plate heat transfer coefficient distributions in figure 7.10. The heat transfer coefficient in this zone using a 4mm profile fan is almost 1.5 times greater than that using the 6.5mm profile fan. Consequently, with the deflection of air from the top plate to the base plate, the approach velocity for the second impingement is also considerably lower when using a 6.5mm profile fan.

This hypothesis is supported by examining the fluctuations in heat transfer coefficient on the base plate in figure 7.9. For the 2mm and 4mm profile fans, a decrease in $\sigma_h$ is observed in the secondary impingement region. An increase in unsteadiness is also observed in the secondary vortex region, particularly for the 4mm profile fan shown in figure 7.9 d). In figure 7.9 f), the fluctuating heat transfer coefficient also increases over a similar annular region, suggesting the existence of a secondary vortex which produces a detachment of the shear flow along the base. In addition, outside this annular region of increased $\sigma_h$, a decrease in the magnitude of the fluctuations in $h_{fc}$ is evident over an annular region implying that some level of secondary impingement or fluid reattachment with the base plate exists but is too weak to result in a significant increase in $h_{fc}$ in figure 7.9 e).
Figure 7.9: Forced convection heat transfer coefficient (left) and fluctuations in $h_{fc}$ (right) on the base plate of a finless heat sink for a $\Omega 15$mm radial fan at 10,000 rpm and profile height of a), b) 2mm; c), d) 4mm; and e), f) 6.5mm. Contour levels: a), c), e) 5W/m²K; b), d) 0.005; f) 0.1.
Figure 7.10: Forced convection heat transfer coefficient (left) and fluctuations in $h_{fc}$ (right) on the top plate of a finless heat sink for a Ø15mm radial fan at 10,000 rpm and profile height of a), b) 2mm; c), d) 4mm; and e), f) 6.5mm. Contour levels: a), c), e) 5W/m$^2$K; b), d) 0.01; f) 0.02.
The distribution of $\sigma_h$ presented in figure 7.9 and figure 7.10 also highlight the considerable increase in surface heat transfer fluctuations when operating outside the maximum fan profile suggested in the previous section. For the 2mm and 4mm fan profiles, the fluctuations in heat transfer coefficient are a maximum of approximately 10%. This is shown to increase to 50% when using a 6.5mm profile fan. The larger spacing between the finless base and top plates, combined with no benefit in flow rate through the channel for $a_r > 0.3$, promotes unsteady vortical fluid motion similar to that shown in figure 7.5 a) and b) but to a greater extent. For a lower aspect ratio fan, although the outlet flow is unsteady, these structures are somewhat suppressed by the mean flow as shown in the instantaneous velocity fields in figure 7.8 a) and b) for $a_r = 0.068$.

Figure 7.11 presents the axisymmetric radial distribution in Nusselt number for varying fan and finless heat sink profiles at a rotational speed of 10,000 rpm using a Ø15mm radial fan. On the base plate, the Nusselt number beneath the fan back plate is similar for all profiles as expected. In this region, it is anticipated that a couette type flow drives heat transfer, and is independent of fan profile for a constant fan diameter and blade Reynolds number. A velocity gradient exists between the finless base plate and the underside of the fan back plate. Although a piezometric pressure difference is absent in this region, this gradient occurs as a result of the moving boundary that is the fan back plate. The 2mm profile fan produces the greatest peak in heat transfer at $r/D = 0.6$, which may be attributed to the increase in acceleration of the fluid upon exiting the blade passage ($\pi D_in H_f < \pi D_{in}^2 / 4$), as previously discussed in section 7.1.1. As profile height increases, the reduced acceleration results in a peak of lower magnitude at $r/D = 0.6$. Bleier (1997) indicated that the fluid also undergoes an acceleration over the blade tip speed, when exiting at the fan blade pressure side. This is confirmed in the heat transfer measurements, as the peak in maximum heat transfer is outside the blade tip location of $r/D = 0.5$. For $r/D > 0.6$, a decrease in $Nu_D$ is experienced as the high momentum fluid exiting the fan detaches from the base plate, due to an adverse pressure gradient, and provides impingement cooling for the top plate at $r/D = 0.7$ for the 2mm profile fan and heat sink. The 4mm and 6.5mm profiles have this peak in $Nu_D$ at $r/D = 0.9$, owing to the increase in flow rate extending the shearing area on the base plate surface.
Upon impingement, the high momentum fluid is then deflected back towards the base plate resulting in the creation of a secondary peak in heat transfer for the 2mm and 4mm profiles over the entire range of Reynolds numbers investigated. For the 6.5mm profile case presented in figure 7.11 a) however, this secondary peak is absent and there is a gradual reduction in heat transfer from the local maximum. The level of heat transfer from the top plate surface in figure 7.11 b) is also greatly reduced over the 2mm and 4mm profiles, as the fluid predominately exits the fan blade along the base plate. Although the 2mm profile fan supplies 57% of $\dot{Q}$ for the 4mm profile fan at this rotational speed, the 2mm profile provides a similar magnitude of radial heat transfer on the top plate as the 4mm profile. It is estimated that the mean velocity within the finless channel is similar for both, as $\approx 40\%$ reduction in flow rate is coupled with a 40% reduction in exit flow area due to the finless plate spacing reducing from 5mm ($H_f = 4\text{mm}$) to 3mm ($H_f = 2\text{mm}$). Therefore fan profile selection can greatly influence the heat transfer performance of both base and top plates of this finless design.

An analogy between miniature radial fan exit flow characteristics and resultant surface heat transfer for base and top plates is presented in figure 7.12 for the 2mm fan profile example. The scaling relationships of figure 7.12 a) and b) are with a Reynolds number which is characterised by fan diameter, $D$, and the mean velocity of the exit flow, calculated
using flow rate measurements. This is consistent with the scaling relationships provided in Chapter 6 for axial fans. Using a 2mm profile fan, $Nu_D \sim Re_D^{0.5}$ for $r/D > 0.25$ of the base plate suggesting that the relatively steady flow as indicated by the magnitude of fluctuations in $h_{fc}$ of figure 7.9 b), dominates the heat transfer. At $r/D \approx 1 - 1.5$, a slight deviation occurs as the secondary time averaged vortex and impingement provide the mechanism for heat transfer in this region. It is also apparent that the secondary peak in heat transfer shifts in the radial direction with increasing Reynolds number. This is less noticeable at higher $Re_D$ as the increased flow rate between the top and base plates tends to squeeze the secondary vortex and promote a closer representation of a parabolic flow between parallel plates. This effect alters the angle at which the air deflects from the top surface to the base, therefore shifting the point of impingement further downstream. This is also evident for $H_f = 4$mm, where the scaling relationship is $Nu_D \sim Re_D^{0.6}$ for the base plate. The scaling relationship for $H_f = 6.5$mm is $Nu_D \sim Re_D^{0.7}$. It is clear that a trend exists between the heat transfer on the base plate and the effect of fan and heat sink profile scaling on fluid dynamics within the finless heat sink. This has also been noted for the local heat transfer measurements in figure 7.9.

The instantaneous velocity field data in figure 7.5 for $H_f = 4$mm shows vortices created by the rotating blades and at the shear layer region travelling both in the radial and axial directions until breakdown. It is this axial movement which allows the vortex to disturb the
shearing flow on the base plate surface. As the distance between the finless plates increase with increases in fan profile, a greater area exists for vortex growth and flow unsteadiness. This is particularly the case for a fan aspect ratio > 0.3, as no benefit in flow rate is achieved above this. Therefore, the flow tends towards the fan back plate, and a large portion of the blade profile does not contribute to the air flow between the finless plates of the heat sink. This has already been shown when comparing the top plate Nusselt number for $H_f = 6.5\text{mm}$ over the other profiles examined in figure 7.11. The increased unsteadiness between the finless plates for larger aspect ratios is reflected in the surface heat transfer with increases in the Reynolds number exponent from 0.5 to 0.7 for the base plate and 0.6 to 0.8 for the top plate. Recalling the bulk heat transfer analyses in Chapter 5, a similar scaling effect with Reynolds number was noted when comparing the larger flow area finless designs with conventional finned designs which directed the flow into a parabolic profile and suppressed unsteady fluid motion. The top plate heat transfer scales to a greater power as a result of the increased unsteadiness in the fluid flow in the top half of the finless channel. The peak in heat transfer due to the impinging flow on the top surface scales with $Re_D^{0.5}$ for all profiles examined.

Recalling the time-averaged velocity fields of figure 7.4, a radial shift in the locations of impingement on the top and base plates is evident with increasing flow rate. This is reflected in the scaling relationships of figure 7.12 as the $r/D$ location of the secondary peak in heat transfer on the base plate increases with increasing Reynolds number. This trend is also noticed for the primary impingement on the top plate.

At the miniature scale, mixed convection can often contribute, both positively and negatively, to the heat transfer distribution (Raja et al., 2009). The influence of mixed convection on the heat transfer performance of the smallest fan and lowest flow rates considered in this thesis are also shown in the scaling relationships presented in figure 7.12. For $Re_D \approx 240$, the heat transfer distribution deviates from the relationship with Reynolds number, for both base and top plates.

The contributions of natural convection to the heat transferred can be investigated through the Richardson number, $Ri_L = Gr_L/Re_L^2$, which defines the ratio of buoyancy to momentum forces. To maintain physical sense, the definition of $Ri_L$ cannot be described through the use
of $Re_D$ as the characteristic length scale, $D$, is not linked with the buoyancy contributions on the flat surface. Therefore the experimental arrangement determines the correct definition of this parameter. The $\varnothing$15mm fans were positioned in the centre of the heated base plate, like shown in figure 7.13. The fan exit flow generates a boundary layer on both the base and top plates which grows along the length $\approx 0.5W$. Similarly, the growth of the boundary layer due to natural convection is from the plate edge to the centre along the base plate. The appropriate length scale for $Ri_L$ is therefore $L = 0.5W$, or half the width of the heated surface, as this governs both natural convection and forced convection modes of transport where boundary layer growth is proportional to the length $0.5W$ for base and top surfaces. For the 2mm profile fan, buoyancy effects dramatically increase from $Ri_L = 0.05$ ($Re_D \approx 600$), to $Ri_L = 9.1$ for the lowest Reynolds number presented in figure 7.12. At $93 < Re_D < 238$, an operating point exists where forced and natural convection contributions are at a similar level ($Ri_L \approx 1$), as the corresponding range of Richardson number is $2.25 > Ri_L > 0.3$. This is also shown graphically in figure 7.12, as the trend in the radial distribution of Nusselt number for $Re_D = 238$ approaches that of the collapsed data for $Re_D > 600$, where buoyancy induced flow has a minor effect on heat transfer. The transition to mixed convection at $Re_D < 600$ agrees for fans examined with $a_r < 0.3$, which accommodate increased flow rates over the $\varnothing$15mm 2mm profile fan. From the definition of the Richardson number, it is clear that varying strength of buoyancy forces will result in changing the point of transition where mixed convection may influence heat transfer. The experiments were carried out for a Rayleigh number range $Ra_W = 10^5 - 10^6$, which would similarly occur in electronics cooling applications, based on the maximum allowable component temperatures which limit reliability. Therefore for the finless cooling solution design presented in this chapter, it is anticipated that buoyancy effects will only begin to influence heat transfer performance.

![Figure 7.13: Defining the Richardson number based on the experimental arrangement.](image-url)
significantly for $Re_D < 600$ in electronics cooling applications.

Figure 7.14 presents the area average Nusselt number for the three fan profiles and a fan rotational speed of 10,000 rpm. The result emphasises the effect of varying the spacing between finless plates with various fan profiles, and also varying flow rate. The 4mm profile fan and channel spacing is most beneficial on heat transfer for $r/D > 1$. The Nusselt number for top and base plates using a 2mm profile fan illustrates that low flow rate designs can produce similar heat transfer rates over a wide $r/D$ range. At $r/D = 1.5$, the average Nusselt number on the base and top plates is 12% and 9% lower when using a fan with $a_r = 0.133$ compared to a fan with $a_r = 0.267$. Using a fan with $a_r = 0.433$ results in reductions of 2.4% and 36%. As discussed previously, the 6.5mm profile design ($a_r = 0.433$) decreases heat transfer over the alternate designs for the top plate surface. The channel spacing, which depends on fan profile, should therefore be considered for $a_r < 0.3$ to prevent this degradation on heat transfer. Figure 7.14 also reiterates the importance of considering the fan and heat sink in combination during the design process. In this example the 2mm profile fan requires lower flow rate and less space to provide a similar level of heat dissipation as the 4mm fan design.

![Figure 7.14: Area average Nusselt number for varying fan profile height at 10,000 rpm.](image-url)
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

Diameter scaling of fan

In this section, the influence of fan diameter scaling is discussed for a constant fan profile of 4mm. The local heat transfer coefficient for the base and top plate are shown in figure 7.15 for fan diameters and rotational speeds of Ø24mm, Ø32mm, and Ø59mm, and 6000 rpm, 1000 rpm, and 1000 rpm respectively. As noted previously in the velocity field analysis for the Ø24mm fan, above approximately 4000 rpm, a secondary vortex no longer exists in the flow field and this is reflected in the local heat transfer distribution presented in figure 7.15 a) and b) for a 6000 rpm example. Heat transfer coefficient magnitudes up to \( \approx 180 \text{W/m}^2\text{K} \) are experienced on the base plate as a result of the large gradients in fan exit flow velocity with the base surface. The heat transfer on the top surface is enhanced by the primary vortex due to the recirculation of ambient temperature fluid along the top plate. In figure 7.16 the standard deviation and minimum fluctuations in heat transfer coefficient on the base plate are presented. Using the Ø24mm fan, the fluctuations in \( h_f, c \) are a similar magnitude to that observed for the Ø15mm fan in the previous section on profile scaling. The minimum fluctuations shown in figure 7.16 b) are up to 30% of the heat transfer coefficient and also highlight the whirling exit flow from the fan blades, which is in the direction of fan rotation. This signifies that the toroidal vortices observed within the finless channel with the velocity field measurements in section 7.1.2 are three-dimensional.

At 1000 rpm, the Ø32mm fan provides an annular region of increased heat transfer coefficient due to the secondary impingement on the base plate observed in the time-averaged velocity field of figure 7.6 a). The fluctuations in heat transfer coefficient (\( \sigma_h \)) are up to approximately 25% and cover an annular region on the surface which is occupied by the secondary vortex. This region of increased unsteadiness is also reflected in figure 7.16 d) for \( \sigma_{h,\text{min}} \).

The distribution of forced convection heat transfer coefficient using a Ø59mm fan operating at 1000 rpm is presented in figure 7.15 e) and f). The trend is similar to that of larger aspect ratio fans examined, however the peak in \( h_f, c \) has become less pronounced due to the smaller area in the flow field occupied by the primary vortex as seen in the time-averaged velocity magnitude for the same rotational speed of figure 7.7 a). As previously discussed, the fluctuations in heat transfer coefficient indicate the whirl angle of the exit flow, and
Figure 7.15: Forced convection heat transfer coefficient on base (left) and top (right) plates of a finless heat sink for a radial fan with $H_f = 4$mm and a), b) $\varnothing24$mm at 6000 rpm; c), d) $\varnothing32$mm at 1000 rpm; and e), f) $\varnothing59$mm at 1000 rpm. Contour levels: a), b) 5W/m$^2$K; c) - f) 3W/m$^2$K.
Figure 7.16: Normalised standard deviation (left) and minimum (right) fluctuations in $h_{fc}$ on the base plate of a finless heat sink for a radial fan with $H_f = 4\text{mm}$ and a), b) $\varnothing 24\text{mm}$ at 6000 rpm; c), d) $\varnothing 32\text{mm}$ at 1000 rpm; and e), f) $\varnothing 59\text{mm}$ at 1000 rpm. Contour levels: a), e) 0.01; b), f) 0.03; c) 0.02; d) 0.05.
this is also shown for the Ø59mm fan in figure 7.16 e) and f). This type of whirling outlet
flow is synonmonous with centrifugal fan designs without a volute and also noted by Hanly
(2009) who examined the exit flow field of a miniature radial fan with a backward curved
airfoil blade design using PIV.

The results presented in figure 7.16 imply a larger amplitude of normalised fluctuations
in heat transfer coefficient exist when this secondary vortex is apparent in the flow field.
However, due to the limitations of the experimental technique in resolving the true ampli-
tude of high frequency fluctuations (section 4.2.2), it is not possible to confidently arrive at
this conclusion. It is however, possible to conclude that fluctuations in heat transfer coeffi-
cient with approximately \( f < 5 \text{Hz} \), are a maximum when this secondary vortex is in the
flow field.

The radial average of the Nusselt number is presented in figure 7.17 for each fan diam-
eter of 24mm, 32mm, and 59mm for a range of Reynolds numbers, \( 645 < Re_D < 34560 \).
The non-dimensional heat transfer data is presented through its relationship with Reynolds
number. A similar trend is observed for all diameters to that presented for the Ø15mm
fan previously. A Reynolds number exponent of 0.65 was selected to describe the simi-
larity in scaling relationship for independent fan diameters and aspect ratios \( a_r < 0.167 \).
The Reynolds numbers where the secondary peak in heat transfer occurs are highlighted
in figure 7.17 a) and c). As shown for the Ø15mm fan, this peak shifts outwards in the
radial direction as Reynolds number is increased, until a point where the secondary vortex
is overcome by the mean radial flow. The shift in the peak in Nusselt number on the top
plate is also discernible in figure 7.17 b), d) and e). This peak becomes less pronounced
as Reynolds number is increased above \( Re_D \approx 13700 \). At this point, it is postulated that
the circulatory flow due to the primary vortex is being dissipated into the mean flow, in a
similar manner to the influence on the secondary vortex for lower Reynolds numbers. This
is supported by the velocity field measurements for the Ø59mm fan at a lower Reynolds
number of \( Re_D = 6520 \), presented in figure 7.7, as the primary vortex is contained in a much
smaller area of the radial-axial plane than for lower rotational speeds.

Scaling of fan diameter for a constant profile height indicates that large diameter fans
can produce high levels of heat transfer enhancement \( (h_{fc} \sim Re_D^{0.65}) \) while maintaining a
Figure 7.17: Nusselt and Reynolds number scaling relationship on base (left) and top (right) plates using a), b) Ø24mm; c), d) Ø32mm; and e), f) Ø59mm radial fans with $H_f = 4\text{mm}$.
low profile. These findings are also useful for larger scale computing systems with densely packed printed circuit boards resulting in limited access for direct component cooling. A low profile radial fan such as the Ø59mm design could be implemented into such environments and achieve local heat transfer rates of over 200W/m²K. The scaling relationships of figure 7.17 for the combined fan - finless heat sink design reiterates the previous findings that the thermal performance of miniature and low profile finless cooling solution designs typically correlate towards that of a turbulent flow regime, despite operating at low Reynolds numbers in many cases. The unsteady fluid motion within the finless channel, described in the previous sections through velocity field and surface heat transfer measurements, provides some insight into the fluidic mechanisms which promote heat transfer for a low profile finless design.

**Comparisons in heat transfer performance**

A number of factors influence the selection of a cooling solution for use in portable and low profile devices. Heat dissipation levels, space requirements, manufacturing cost, and power usage are generally the primary factors in design choice. This thesis presents local heat transfer distributions utilising axial fan impingement and radial fan cooling to augment thermal performance. As both methods comprise of a finless surface for convection, the main difference is in the fan design and it is assumed that manufacturing costs are on par for both. Heat transfer coefficients are also at a similar level, however the relative thermal performance for portable and low profile devices should be considered with respect to footprint area, cooling solution profile, and power usage. Therefore a comparative analysis of the heat transfer performance between both methods of cooling is presented in this section.

The relative performance between both designs was considered by comparing thermal performance with varying pumping power. In order to confidently measure the power requirement without any bias, both fan types were operated using the same motor (Maxon 110049 24VDC) for the purpose of power measurement. Fan power was then evaluated, accounting for the unloaded motor power requirement, using the relationship between motor torque and rotational speed, \( P = T \omega \).
In figure 7.18, the area averaged heat transfer coefficient using an axial fan with 24.6mm diameter and a fan to plate distance of 5mm has been compared to 15mm and 24mm diameter radial fans with profile heights of 4mm. The area average is the average heat transfer coefficient within a circular area centred on the fan central axis, with radius \( r \). The results for the axial fan, presented in figure 7.18, are for a nominal rotational speed of 9000 rpm, for the practical case of inlet mounted motor supports and the ideal case without any motor supports based on the findings in section 6.2. The heat transfer performance using a Ø15mm radial fan is shown when operating at 50% of the power required by the axial fan. In addition, the average heat transfer performance is also presented in figure 7.18 using a Ø24mm radial fan design operating at the same power required by the axial fan and a similar fan footprint area.

The average heat transfer coefficient is similar for the axial and Ø15mm radial fans up to \( r = 0.005 \text{m} \). This suggests that the recirculating flow beneath the axial fan hub at this height from the plate does not promote heat transfer above the couette type flow experienced beneath the radial fan back plate. This is mainly due to the close proximity of the axial fan with the surface, which suppresses the circulatory effects as shown in a velocity field study by Sui et al. (2009a). The area averaged heat transfer coefficient for the practical inlet mounted motor support arrangement is within 5% of the same for the Ø15mm radial fan for \( r > 0.0245 \text{m} \). At lower radial locations, the Ø15mm radial fan achieves up to 26% enhancement over the axial fan with motor supports. In contrast, the axial fan design
without motor supports produces an enhancement of up to 10% over the Ø15mm radial fan for \( r > 0.0197 \text{m} \). However, the radial design provides up to 16% increase over this ideal axial fan arrangement for \( 0.006 \text{m} < r < 0.0197 \text{m} \).

The maximum heat transfer coefficient for all designs compared in figure 7.18 is achieved using the Ø24mm radial fan operating at the same power input as the axial fan design. An almost constant enhancement of 25% is experienced for \( 0.025 \text{m} < r < 0.04 \text{m} \) over the ideal axial fan design without motor supports. Substantial increases in \( \dot{h}_{fc} \), however, are seen for \( 0.005 \text{m} < r < 0.01 \text{m} \) when using this axial fan design over the Ø24mm radial fan. This is expected, as the radial fan blade tip is only at \( r = 0.012 \text{m} \).

It can be concluded therefore that the radial fan designs are preferable to the axial fan arrangement. The Ø15mm radial fan has a lower footprint area and can operate at 50% of the power required by the axial fan design yet dissipate heat at a similar and often increased rate. The Ø24mm radial fan was shown to promote heat transfer over the axial design for an equal power requirement. Finally, the profile of the radial fan arrangement is just one third of the axial fan cooling solution profile. Consequently, the radial fan design has been considered to examine the effect of practical operating conditions on surface heat transfer.

**Influences of fan inlet confinement**

In many applications, space constraints commonly result in cooling solutions being densely packed within an enclosure to maintain an overall compact device design. In these instances, the inlet flow to the fan can be restricted and adversely influence the outlet flow which augments heat transfer. Therefore it is necessary to examine the effect of inlet confinement on heat transfer performance over the same cooling solution arrangement without confinement. The Ø59mm fan was considered for this analysis as it has the greatest confinement ratio \( D/H_{CF} \) of all the fan designs examined, consequently imposing the greatest resistance to fan flow rate.

Figure 7.19 presents the relative decrease in local heat transfer coefficient over an unconfined condition for confinement plate heights from the inlet, \( H_{CF} \), of 26.8mm and 6.7mm with a fan rotational speed of 2000 rpm. Decreasing \( H_{CF} \) has a relatively minor effect on the local heat transfer distribution for \( D/H_{CF} = 2.2 \), however it does degrade it by up to
20% for $D/H_{CF} = 8.8$ and $r/D \approx 1$. This is due to a decrease in flow rate through the fan with the added resistance of a blockaging effect at the inlet. In the region beneath the fan blades, heat transfer rates remain almost unchanged. The presence of a confinement plate forces the flow to turn 180° as opposed to 90° from the inlet to outlet of the fan blades. Velocity field measurements at the inlet confirm this, and shall be discussed in the proceeding section on the influences of crossing air flows on thermal performance. The majority of the inlet air flow tends towards the inlet orifice edge where the first 90° turn of air into the blade passages exists due to the pressure gradient produced by the rotating blades. The second 90° turn directs the air in a radial outward direction from the pressure side of the blades. Although flow rate is reduced leading to an overall reduction in heat transfer downstream, the fan blade velocity remains unchanged, and therefore the velocity gradient beneath the floating blades is maintained. The variation beneath the fan back plate is minor, as the average heat transfer coefficient in this region is $\approx 18\text{W/m}^2\text{K}$.

![Figure 7.19](image1.png)

Figure 7.19: The influence of fan inlet confinement on the local heat transfer coefficient for a) $D/H_{CF} = 2.2$ and b) $D/H_{CF} = 8.8$.

### 7.1.4 Summary

In the preceeding sections, a number of parameters for a radial finless design were varied, and the resultant effects on fluid dynamics and heat transfer performance were assessed. Velocity field measurements of diameter scaled radial fan designs from 15mm to 59mm indicated unsteady fluid structures in the finless channel which resulted in increased turbulence. High levels of turbulence intensity and Reynolds stresses were attributed to the
interaction of vortices with the high shear flow that exited the fan blade along the base plate. As fan aspect ratio decreased and flow rate increased, the time-averaged toroidal vortices observed in the radial-axial flow field appear to dissipate into the mean radial flow. This resulted in a flow profile which began to resemble a structured parabolic shape downstream of the fan outlet. The complex velocity fields at the near fan exit flow region are graphically summarised in figures 7.20 and 7.21 for the maximum and minimum fan diameters of 15mm and 59mm with profile height of 4mm.

The effects of some fluidic mechanisms observed in the flow field analysis were also distinguished in the local surface heat transfer measurements. Local heat transfer coefficient distributions highlighted an increase in thermal performance on the base plate which directly resulted from the high shear flow at the fan exit. An annular peak in heat transfer on the top plate was found to correspond with a primary flow impingement along with the circulatory flow of a time-averaged vortex in the flow field. A secondary vortex in the finless channel opposed the shear flow and reduced heat transfer over an annular region on the base plate while simultaneously increasing heat transfer fluctuations. This band of low heat transfer however, was also followed by an increase for \( a_r < 0.3 \) due to a secondary impingement. The heat transfer measurements also illustrated that the flow field within the finless channel is three-dimensional due to the centrifugal forces in the fan exit flow. It was noted that the profile of a finless channel should be limited to a maximum fan aspect ratio of \( a_r = 0.3 \) to avoid substantial degradation in heat transfer.

A thermal performance comparison was made between miniature axial fan impingement from Chapter 6, and the miniature radial fan designs examined in this chapter. Ø15mm and Ø24mm radial fans were chosen to operate at 50% and 100% of the power required by the axial fan. It was determined that the radial fan designs were the preferred design choice for thermal management of portable devices based on cooling solution space and power usage requirements. The radial fan and finless channel design has therefore been selected to investigate a practical situation of crossing air flow above the fan inlet. Fan inlet confinement has been shown to reduce local heat transfer performance for \( r/D > 0.5 \) due to a reduction in flow rate. The second part of this chapter examines the combined influence of fan inlet confinement and cross flow for the smallest and largest fan diameters of 15mm
and 59mm with profile height $H_f = 4\text{mm}$, considered here. The maximum and minimum diameter fans were chosen to determine if fluid dynamic and heat transfer trends were common for miniature scale designs with different dimensions and aspect ratios. The graphical summary of the flow field and heat transfer distributions for both these fan designs without a cross flow in figures 7.20 and 7.21 shall be used as a reference in the proceeding section for the discussion.
Figure 7.21: Summary of the velocity field and surface heat transfer distributions using a Ø59mm fan with \( H_f = 4\text{mm} \) operating at 1000 rpm.
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

7.2 Radial fans and finless heat sink with a crossing air flow

In this section, the influence of a crossing air flow over the fan inlet, on heat transfer performance is considered for two fan diameters and a constant fan profile height of $H_f = 4\text{mm}$. Various cross flow magnitudes above the fan inlet were chosen to determine the indirect influence of a crossing airflow on fluid dynamic and heat transfer distributions within the finless design. This thesis does not examine the introduction of a crossing air flow between the finless plates, which would directly influence the fan outlet flow. The two fans of diameter 15mm and 59mm are the minimum and maximum diameters examined in the previous section. These were chosen to investigate if heat transfer trends were common for this design despite dissimilar dimensions. A fan profile height of 4mm was selected for both designs to span the range of aspect ratios for $a_r < 0.3$ examined previously. The Ø15mm fan was designed to rotate counterclockwise whereas the Ø59mm fan was designed for clockwise rotation. Both fan designs remained forward curved during operation. This was considered to investigate any dependence of heat transfer trends on rotational direction.

In section 4.4, the local heat transfer and velocity field measurement planes are presented. These planes have been labelled in figure 4.34 and shall be referenced in the current section for clarity in the discussion of results. A crossflow generator was designed to provide uniform cross flows for a range of velocities and fan inlet confinements. The characterisation of this crossflow generator is presented in section 4.5, which includes the range of velocities examined for both fans. The maximum cross flow velocity at the smallest fan inlet confinement height ($U_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$) has been selected for the majority of graphical presentations in this section. This test case was chosen as it resulted in some of the largest variations in the cooling solution performance for both 15mm and 59mm diameter fans. Hence, most observations for the remaining cross flow velocities and confinement heights in table 4.6 are encompassed in the discussion of this test case.

Velocity field measurements of the flow within the finless heat sink and approaching the fan inlet are presented first. This is followed by an examination of local heat transfer distributions on the base and top finless plates. The flow structures within the finless design,
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

under the influence of a cross flow, are shown for three radial-axial planes positioned 90° apart and compared to the axisymmetric flow fields presented in the first part of this chapter in section 7.1.2. Velocity field measurements were recorded above the fan inlet to assess the distortion of the fan inlet flow due to cross flows. In addition, the local heat transfer distribution behind the fan inlet on the top surface of the top plate, is also discussed for this purpose. Convection on this surface is directly due to the crossing air flow. Therefore, the flow phenomena from the interaction between the cross flow and fan inlet flow are reflected in the heat transfer measurements of this surface.

The significance of a cross flow on the positioning of discrete heat sources relative to a radial fan outlet is discussed using the local heat transfer distributions on the base plate. Recommendations for component positioning are outlined to maximise the component heat transfer coefficient. Finally, circumferential and area averages of the heat transfer performance in the finless heat sink under the influence of various crossing air flows are presented. This is then compared with the same arrangement without crossing air flows or fan inlet confinement.

7.2.1 Velocity field

The time-averaged velocity magnitude and streamlines in the radial-axial region V1 is presented in figure 7.22 for the Ø15mm fan and rotational speeds of 1000 - 10,000 rpm. This velocity field is the result of a cross flow of 1.48m/s with $H_{CF} = 6.7$mm. Compared to the axisymmetric velocity field presented in figure 7.20 for a corresponding fan rotational speed without any cross flow, large differences in the flow field are apparent. Although some curvature in the streamlines exist in the upper half of the finless channel, the primary vortex is no longer dominant in this time-averaged flow field under the influence of a crossing air flow. There appears to be a small vortical motion near the top plate at 10,000 rpm, however it occupies a fraction of the area of the primary vortex shown in figure 7.20 which dominates approximately half of the channel spacing near the fan exit. Aside from the noticeable increase in velocity magnitude, the high velocity shear flow is extended in the radial direction. This high velocity exit flow then tends towards the top plate, similar to the case without cross flow. However, it does so in the absence of a secondary vortex
Figure 7.22: Time-averaged velocity magnitude and streamlines using a Ø15mm fan with $H_f = 4\text{mm}$ operating at a) 1000 rpm, b) 5000 rpm, and c) 10,000 rpm under the influence of a crossing air flow of $U_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Region V1 presented.

which was shown to produce the adverse pressure gradient.

The instantaneous velocity fields of figure 7.23 a) and b) indicate less vortices in this region compared to the instantaneous flow field in figure 7.20, particularly up to 0.02m from the fan blades. However, the fan exit flow remains unsteady. The radial turbulence levels $I_r$, shown in figure 7.23 c), are at a similar level as without any cross flow. There is a minor increase observed in the shear layer produced as the shearing flow rises from the base towards the top plate. The axial turbulence intensity $I_z$, is decreased in the shear layer of the high velocity fan exit flow, presented in figure 7.23 d). This can be attributed to the reduction in vortical structures described previously through the instantaneous and time-averaged velocity magnitudes. Without a cross flow, these vortices interact with the high velocity fan exit flow, therefore increasing axial turbulence levels. Additionally, Reynolds shear stresses due to acceleration and deceleration in the flow field were typically 25% lower in this plane when under the influence of cross flow.

In the plane V2, a further difference in the velocity field is observed. This is shown in figure 7.24 for time-averaged velocity magnitude and Reynolds shear stress, and a fan rotational speed of 10,000 rpm. In this region, both primary and secondary vortices exist in
Figure 7.23: Velocity field statistics within a finless heat sink design using a Ø15mm fan with \( H_f = 4\text{mm} \) operating at 10,000 rpm and under the influence of a crossing air flow of \( \overline{U}_{CF} = 1.48\text{m/s} \) and \( H_{CF} = 6.7\text{mm} \). a), b) Instantaneous velocity magnitudes at 0s and +1s; c) Turbulence intensity in the radial direction; d) Turbulence intensity in the axial direction; and e) Reynolds shear stress. Region V1 presented.
addition to other low velocity vortices downstream. Compared to the velocity field of figure 7.22 c), it is possible to conclude a highly three-dimensional fan outlet flow results from the crossing air flow over the fan inlet. This is also confirmed with velocity field measurements of region V3, presented in figure 7.25. The existence of these three-dimensional vortex structures in the flow field also results in the high velocity air flow deviating from the base plate to the top plate noted previously for region V1 in figure 7.22. In figure 7.24 b), an elevated magnitude of Reynolds shear stress is evident along the shear layers of the high velocity fan exit flow. Here, the interaction of large vortical structures with the high velocity exit flow produces increased levels of acceleration and deceleration. This is also apparent in region V3, as shown in figure 7.25 b).

The velocity field of region V3 in figure 7.25 resembles that of the Ø24mm fan for fan speeds above 4000 rpm (figure 5.14) where the adverse pressure gradient due to the secondary vortex is overcome by an increase in mean flow through the finless channel. It is anticipated that the same has occurred here for the Ø15mm fan due to a substantial increase in flow rate through fan blades at this circumferential location by the crossing air flow. Consequently flow rate is increased through the finless channel in this region, without increasing fan speed.

The velocity field analysis of the planes V1 - V3, suggests that certain circumferential
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

Figure 7.25: Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm and under the influence of a crossing air flow of $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. Region V3 presented.

Fan exit flow positions display increased performance in the radial-axial direction (V1 and V3), whereas other positions show a degradation in performance over the case without any cross flow above the fan inlet (V2). This can be attributed to the pressure variation above the fan inlet and distortion of the fan inlet flow due to the velocity magnitude and direction of the crossing air flow.

In figure 7.26, velocity field data are presented for the measurement plane V5, 2mm above the fan inlet, and cases of with and without a cross flow. In figure 7.26 a), the inlet flow and pressure, is evenly distributed about the fan inlet orifice, resulting in the axisymmetric velocity fields and surface heat transfer distributions discussed in the previous section. By introducing a cross flow, the fan inlet flow becomes distorted from the original radial inflow to a swirling inflow. A saddle point is created behind the fan inlet where the crossing air flow directly opposes the fan inlet flow. At this point, the greatest resistance to the fan inlet flow is apparent, and it divides the streams which approach the inlet orifice with the streams which continue in the direction of the cross flow. This arrangement is similar to that investigated by Holdo et al. (2000) for flows generated by the interaction of a circular inlet and a cross flow. Based on flow patterns visualised using a surface oil-film technique, the generic patterns in the flow field about the circular inlet are shown in figure 7.27. The
Figure 7.26: Time-averaged velocity magnitude and streamlines a) without a cross flow, and b) with a cross flow of $U_{CF} = 2.06\text{m/s}$, in the vicinity above the inlet (V5) of a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm.
Figure 7.27: Surface shear stress pattern for inlet flows with a cross flow, replotted from Holdo et al. (2000).

Figure 7.27: Surface shear stress pattern for inlet flows with a cross flow, replotted from Holdo et al. (2000).

differences between that presented in figure 7.26 b), for the current experimental investigations, and that described by Holdo et al. (2000) for a solely axial suction velocity, highlight the centrifugal forces at the inlet for fan flows with a crossing air flow. The swirling inflow results in only one stagnant point in the flow field at the rear of the fan inlet. This inflow pattern has a large influence on the outlet flow within the finless heat sink, as shown with the previous velocity field measurements in this section. The low velocity region at the rear of the inlet orifice ($r/D = 0$) and towards the right of the fan inlet ($0 < r/D < 0.5$) indicate the large change in direction the fluid must undergo. This results in a local increase in static pressure which shifts the operating point of the fan locally. Consequently, fan performance is reduced and an outlet flow similar to that observed in figure 7.24 is apparent. However, at the leading edge ($r/D = 0$) and to the left of the fan inlet ($-0.5 < r/D < 0$), fan performance is maintained or even increased locally for regions where the crossing air flow imposes no resistance to the inlet flow and static pressure is reduced. The resultant exit flow fields are similar to that shown in figure 7.22 and figure 7.25. Interestingly, the degrading effects of a cross flow at the fan inlet are realised on a different radial-axial plane at the fan outlet. It is postulated that this rotational shift is due to the centrifugal nature of both the swirling inlet
flow, and whirling outlet flow from the fan.

In addition to local changes in operating point for the fan due to these pressure effects, the inlet flow observed in figure 7.26 b) would also appear to approach the fan blades at an off-angle compared to the standard case without cross flow in figure 7.26 a). The inlet velocity approach angle to fan blades has been documented in particular for rotor-stator arrangements where guide vanes are designed to align the inlet flow with the fan blade for increased performance and multistage arrangements (Bleier, 1997). It is anticipated that the cross flow direction has a major role in this as it can either guide or distort the approaching inlet velocity to the fan blades. Wright et al. (1984) examined the effect of centrifugal fan performance with a distorted inflow, produced by a non-uniform velocity profile over a much larger scale 762mm diameter centrifugal fan with volute. Velocity variations over the inlet of just 10 - 15% resulted in reductions in efficiency and pressure rise of 5%. Furthermore, the authors also found that even moderate levels of distortion reduced the overall fan performance.

Figure 7.28 presents the time-averaged inlet flow field of region V4, directly above plane V1, for the Ø15mm fan standard case without cross flow and also with a cross flow for the same rotational speed of 10,000 rpm. The air flow is directed 180° from inlet to exit due to the inclusion of a confinement plate above the fan inlet. In figure 7.28 b), the flow phenomena discussed for the plan view of the inlet in figure 7.26 b) are also evident in this side profile. The stagnant point in the flow field where the fan inlet flow is opposed by the crossing air flow is apparent. The stagnation pressure in this region results in a reduction of the inlet velocity over the case without cross flow. On the other side of the inlet, the velocity magnitude increases as the cross flow has a positive contribution to the fan inlet flow. Here, the static pressure is at a minimum. Although a local maximum in fan inlet flow resistance occurs in this inlet plane, the outlet plane V1 shows an increase in mass flow rate over that observed in figure 7.28 a). This is due to the centrifugal nature of both the inlet and outlet flow from the fan, previously highlighted in velocity field measurements of V5 (figure 7.26 b)) and heat transfer measurements in the previous section for without a cross flow. Flow exits the fan blades with a tangential component, therefore the degrading effects on fan performance due to this stagnation pressure at the inlet are realised in a different
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

Figure 7.28: Velocity field between base and top plates (V1), and above the fan inlet (V4), for a Ø15mm fan with $H_f = 4\text{mm}$ operating at 10,000 rpm. A fan inlet confinement of $H_{CF} = 6.7\text{mm}$ is presented a) without a cross flow and b) with a cross flow of $\overline{U}_{CF} = 1.48\text{m/s}$.

The effect of cross flow and confinement distance $H_{CF}$, on the velocity field of V1 within a finless heat sink when using a Ø59mm fan at 1000 rpm is presented in figure 7.29 and figure 7.30. Both flow fields are quite similar qualitatively, to the original velocity field summarised in figure 7.21 without a cross flow. This suggests that the crossing air flow levels investigated have a lesser effect on the larger Ø59mm fan performance. However, the time-averaged velocity magnitude in this plane for $\overline{U}_{CF} = 1.96\text{m/s}$ and confinement of 26.8mm is higher than that when $\overline{U}_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$. The increase in velocity magnitude in this plane is again due to a local decrease in static pressure from the assistance of the crossing air flow. Similarly, this effect also results in increased Reynolds
shear stress levels at the fan exit.

Both velocity fields presented in figure 7.29 and figure 7.30 are also presented in figure 7.31 in addition to the inlet velocity field V4 for each confinement height. The inlet velocity fields differ considerably by changing the fan inlet confinement distance, $H_{CF}$ from 26.8mm to 6.7mm. In figure 7.31 a), the crossing air flow above the fan inlet almost maintains a uniform streamwise direction, in the presence of the fan operating at 1000 rpm. There are some differences however, which have been highlighted. The inlet velocity of

![Figure 7.29](image1)

Figure 7.29: Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø59mm fan with $H_f = 4$mm operating at 1000 rpm and under the influence of a crossing air flow of $U_{CF} = 1.96$ m/s and $H_{CF} = 26.8$mm. Region V1 presented.

![Figure 7.30](image2)

Figure 7.30: Time-averaged statistics of a) velocity magnitude and b) Reynolds shear stress using a Ø59mm fan with $H_f = 4$mm operating at 1000 rpm and under the influence of a crossing air flow of $U_{CF} = 1.48$ m/s and $H_{CF} = 6.7$mm. Region V1 presented.
the fan results in pulling the mean cross flow towards the top plate surface, where a minor impingement occurs at the inlet orifice edge. A new boundary layer begins to grow along the top surface of the top plate from this point. The suction effect on the mean cross flow is also experienced at the confinement plate surface above the fan inlet, where the velocity gradient at the wall is reduced. Moving the confinement plate to 6.7mm above the fan inlet results in greater unsteadiness in the air flow above the fan inlet. In this region, a saddle point appears similar to that observed for the Ø15mm examples combined with a number

![Figure 7.31: Velocity field between base and top plates (V1), and above the fan inlet (V4), for a Ø59mm fan with $H_f = 4mm$ operating at 1000 rpm. Fan inlet confinement and cross flow velocity of a) $H_{CF} = 26.8mm$ and $\overline{U}_{CF} = 1.96m/s$, and b) $H_{CF} = 6.7mm$ and $\overline{U}_{CF} = 1.48m/s$ are presented.](image)
of time-averaged vortices. Based on the inlet velocity field measurements presented it can be concluded that pressure variations exist due to the crossing air flow. The distortions to the inlet flow can also degrade the local performance of the fan and therefore the local heat transfer within the finless channel, which shall be discussed in the following section.

7.2.2 Local and average surface heat transfer

The variation in the velocity field around the radial finless heat sink when a crossing air flow is introduced has been discussed. In this section, the resultant surface heat transfer trends are examined, and compared with the observations in the previous section, in addition to that presented for the thermal performance without a cross flow in the earlier sections of this chapter. Firstly, the distribution of local heat transfer coefficient is considered for both Ø15mm and Ø59mm fan arrangements. Recommendations are proposed for improvement of component thermal performance when a radial fan is under the influence of a crossing air flow. Circumferential and area averages of the heat transfer performance with and without cross flow are then compared.

In figure 7.32, the base plate heat transfer coefficient is presented for the Ø15mm fan at rotational speeds of 1000 - 10,000 rpm under the influence of a cross flow with $U_{CF} = 1.48 \text{m/s}$ and $H_{CF} = 6.7 \text{mm}$. The top plate heat transfer coefficient is also shown over an area behind the fan inlet. Although the heat transfer distribution will vary on the top surface around the fan inlet based on the velocity field analyses in the previous section, this region was chosen as it encompasses the performance degrading effect of the high resistance to the fan inlet flow shown in figure 7.26 b). For all local heat transfer distributions presented, fan rotational direction is provided, and the direction of the uniform cross flow is from top to bottom of the figure.

The PIV planes V1 - V3 are highlighted in figure 7.32 a) for a 1000 rpm sample speed. All rotational speeds indicate the surface heat transfer distribution is no longer axisymmetric under the influence of a cross flow $U_{CF} = 1.48 \text{m/s}$. Regions of high heat transfer on the base plate are also positioned directly below the low velocity inflow observed in figure 7.26 b). Conversely, low surface heat transfer regions are located below the high velocity
Figure 7.32: Forced convection heat transfer coefficient on base (left) and top (right) plates of a finless heat sink for a Ø15mm radial fan with $H_f = 4$mm operating at a), b) 1000 rpm, c), d) 5000 rpm, and e), f) 10,000 rpm under the influence of a crossing air flow of $U_{CF} = 1.48$ m/s and $H_{CF} = 6.7$mm. Contour level: 5 W/m²K.
inflow, where the cross flow positively contributes to the fan flow rate. This is in agreement with the velocity field measurements of planes V1 - V3. Figure 7.22 and figure 7.25 demonstrate an increase in flow rate through the finless channel at these circumferential positions over the original time-averaged flow field without a cross flow. Figure 7.24 however, illustrates a reduction in the radial distance of the high shear flow which is reflected in the local heat transfer distributions on the base surface. Despite the loss of the secondary peak in heat transfer with the introduction of a cross flow, it is apparent that some peaks occur outside the high heat transfer region on the base plate shown in figure 7.32. These peaks confirm the existence of three-dimensional fluid structures in the flow field that are no longer axisymmetric.

A mirrored trend of the surface heat transfer along the base plate is observed in figure 7.33 a) and c) for the lower fan speeds of the Ø59mm fan which operates in the opposite rotational direction to the Ø15mm fan. It can be concluded therefore, that the regions of high/low heat transfer are offset in the direction of fan rotation. This is due to the centrifugal effects of both the swirling inflow, and whirling outflow of the fan discussed in the previous section on the velocity fields. The inflow to the fan swirls with fan rotation, and enters the blade channels where it is directed tangentially outwards by the pressure side of the forward curved blade. Both the inlet and outlet flows encompass a tangential velocity component which results in the offset heat transfer coefficient distribution observed.

In figure 7.34, the time-averaged and minimum fluctuations in the heat transfer coefficient are presented for the base plate using both fan diameters. The fluctuating outlet flow highlights the direction of the absolute velocity as it leaves the fan blade. As shown for the Ø59mm fan, the angle of the exit flow, \( \alpha \), is reduced over that observed for the case without cross flow in figure 7.16. A similar reduction in exit flow angle due to an increase in system resistance was also observed in Chapter 5 for the analysis of flow alignment in miniature heat sinks. The lower fluctuations in \( h_{fc} \) correspond to the regions of high surface heat transfer. This is due to a positive contribution by the cross flow which leads to an increase in flow rate as the fan operates locally in a low pressure rise region of the fan performance curve. The higher time-averaged fluctuations are up to 25% of \( h_{fc} \) and correspond to the regions of low surface heat transfer. In this region, the fan operates locally
Figure 7.33: Forced convection heat transfer coefficient on base (left) and top (right) plates of a finless heat sink for a Ø59mm radial fan with \( H_f = 4 \text{mm} \) operating at a), b) 1000 rpm, c), d) 2000 rpm, and e), f) 4000 rpm under the influence of a crossing air flow of \( U_{CF} = 1.48 \text{m/s} \) and \( H_{CF} = 6.7 \text{mm} \). Contour level: 5W/m\(^2\)K.
Figure 7.34: Normalised standard deviation (left) and minimum (right) fluctuations in $h_{fc}$ on the base plate of a finless heat sink for a radial fan with $H_f = 4$mm and a), b) Ø15mm at 10,000 rpm; e), f) Ø59mm at 1000 rpm. Contour levels: a), c) 0.01; b), d) 0.03.

in a high pressure rise region of the fan performance curve due to the opposing effect of the cross flow. This adverse effect on fan performance was often found to result in rotating stall. Local stall has a significant effect on heat transfer as seen in figure 7.33 a) in particular. An increase in $h_{fc}$ to approximately 40W/m$^2$K is only achieved beneath the fan blades due to the gradient produced by rotating blades above the surface. Outside this region the heat transfer coefficient drops off to $h_{fc} \approx 5$W/m$^2$K as a zero flow condition from the fan is apparent. Hence, in addition to increasing fan performance locally, a cross flow can act as a local blockage to the fan inlet flow. The influence the cross flow has on providing a local blockage to the inlet flow is dependent on the magnitude of cross flow and the fan performance attributes. In figure 7.33 e), the Ø59mm fan appears to have overcome any significant stalling effects at a rotational speed of 4000 rpm, and the heat transfer on the base plate returns to an axisymmetric distribution.

The top plate heat transfer coefficient varies considerably compared to the case without
cross flow (figures 7.20 and 7.21) due to the change in fan exit flow profile and also the additional heat dissipating effect of the crossing air flow on the top surface of the top plate. Therefore, despite the degrading effect of rotating stall on the heat transfer distribution within the finless channel, the crossing air flow is shown to increase the heat dissipation levels on the top plate. In figure 7.32 and 7.33, the maximum local heat transfer coefficient on the top plate increases by up to five fold over the maximum local heat transfer coefficient on the top plate without cross flow. This will only be recognised when the convective top surface of the top plate is open to a crossing air flow environment.

In figure 7.32 b), two regions of increased heat transfer coefficient emerge at either side of the fan inlet. These regions of enhancement appear as the fan inlet flow acts as a suction to the crossing air flow. This suction effect has been shown in the previous section (figure 7.31 a)) to result in the growth of a new boundary layer along the top surface in some situations, which enhances heat transfer. The influence of the fan outlet flow on heat transfer begins to appear at the higher fan speeds of 5000 rpm and 10,000 rpm in figure 7.32 d) and f). Regions of increased heat transfer coefficient emerge similar to that observed for the case without cross flow in figure 7.20.

Although the heat transfer coefficient on the top plate increases with increasing rotational speed for the cases presented in figure 7.32 using the Ø15mm fan, a different trend is observed when comparing the three top plate examples for the Ø59mm fan in figure 7.33. For the three fan speeds presented, the intermediate speed of 2000 rpm provides the lowest magnitudes of heat transfer coefficient for the top plate region examined. The reason this occurs becomes clear when separating the contribution of the crossing air flow to the heat transfer on the top plate. The heat transfer coefficient distributions and fluctuations are presented in figure 7.35 for the top surface of the top plate which correspond to the experimental cases shown in figure 7.33.

At 1000 rpm, there is an increase in heat transfer coefficient behind the fan inlet. As briefly discussed using velocity field measurements, this results from the suction of the fan inlet promoting the growth of a new boundary layer along the top plate at the rear of the inlet and increases heat transfer. The use of suction, or a transversal velocity, in boundary layer flows to generate such increases in heat transfer observed in figure 7.35 a), has been
Figure 7.35: Forced convection heat transfer coefficient (left) and fluctuations in $h_{fc}$ (right) on the top surface (fan inlet side) of a finless heat sink for a Ø59mm radial fan with $H_f = 4$mm operating at a), b) 1000 rpm, c), d) 2000 rpm and e), f) 4000 rpm under the influence of a crossing air flow of $\bar{U}_{CF} = 1.48$m/s and $H_{CF} = 6.7$mm. Contour levels: a), c), e) 2W/m$^2$K; b), d), f) 0.01.
presented by a number of authors (Bejan and Kraus, 2003; Ali, 1995; Ahmad et al., 1995).

As fan speed increases to 2000 rpm, the fan static pressure is at a level where the fan inlet
flow begins to revert back to that observed without cross flow, illustrated in figure 7.26 a)
for the Ø15mm fan example. At 4000 rpm, the increased static pressure generated by the
fan overcomes the opposing cross flow effect at the rear of the inlet. Hence, the growth of a
new boundary layer on the top surface no longer occurs as the flow is reversed and the heat
transfer coefficient is reduced in the region behind the fan inlet. This is indicated in the heat
transfer coefficient distributions of figure 7.35 c) and e). The occurrence of this phenomena
is also shown for the Ø15mm fan in figure 7.36 when fan speed is changed from 5000 rpm
to 10,000 rpm at a cross flow velocity $\overline{U}_{CF} = 0.21\text{m/s}$ and $H_{CF} = 6.7\text{mm}$.

Previous studies by Oyewola (2006), Oyewola et al. (2003) and Oyewola et al. (2007)
have examined the influence of transversal velocity from a slot on turbulence statistics. It
was determined that turbulence levels were reduced when the growth of a new boundary
layer was apparent. This is also confirmed when considering the fluctuations in heat trans-
fer coefficient of figure 7.35 b), d) and f). Lower turbulence levels of $\sigma_h < 4\%$ exist behind
the fan inlet when the suction effect of the fan causes renewed boundary layer growth.

Comparing the base plate heat transfer coefficient for the Ø59mm fan in figure 7.33 with
the corresponding top surface heat transfer distribution in figure 7.35, an axisymmetric heat

Figure 7.36: Forced convection heat transfer coefficient on the top surface (fan inlet side) of
a finless heat sink for a Ø15mm radial fan with $H_f = 4\text{mm}$ operating at a) 5000 rpm and b)
10,000 rpm under the influence of a crossing air flow of $\overline{U}_{CF} = 0.21\text{m/s}$ and $H_{CF} = 6.7\text{mm}$.
Contour level: $2\text{W/m}^2\text{K}$. 

transfer distribution occurs when the fan inlet flow overcomes the resistance of the crossing air flow. This emanates as the static pressure of the fan is greater than the opposing pressure caused by the cross flow. Consequently, the fan begins to operate in its recommended operating region, void of any rotating stall. As previously discussed for the lower fan speed of 1000 rpm, a lower static pressure than that produced by the crossing air flow forces the fan to operate in a zero flow region resulting in local stall and an asymmetric heat transfer distribution. In all experiments carried out for both fan diameters, a change in flow direction over the top surface as observed between figure 7.35 b) and figure 7.35 f) resulted in a transition from an asymmetric to an axisymmetric heat transfer distribution. This data is collectively represented in figure 7.37 for both fan diameters of 15mm and 59mm and the range of inlet to cross flow velocities considered, where $U_{in}$ is the mean fan inlet velocity for the cases without a cross flow or confinement. Three fan rotational speeds were examined with nine cross flow velocities, leading to 27 different test cases for each fan diameter. The transition from an asymmetric to an axisymmetric heat transfer distribution was noted when a variation in circumferential heat transfer coefficient about the fan central axis was <12% of the mean. This value was chosen as it is slightly above the uncertainty in the heat transfer coefficient measurements (10.9%).

As shown previously, a cross flow can have a major effect on the local performance of the radial fan. This is reinforced when comparing the cross over to an axisymmetric heat transfer distribution for both fans. The Ø15mm fan produced similar magnitude inlet velocities, $U_{in}$, as the Ø59mm fan for the range of speeds considered in environments without any cross flow disturbance. However, due to the lower pressure rise generated by the miniature Ø15mm fan, the introduction of a cross flow results in a local impedance to the inlet flow, forcing the fan to operate towards a local stalling point which causes the asymmetry in heat transfer distribution. High rotational speeds are therefore required to overcome the adverse effect of cross flows up to approximately 2m/s considered here. Despite operating at similar fan inlet velocities, the Ø59mm fan produced a larger pressure rise ($\Delta P \sim \rho \omega^2 D^2$) to overcome the opposing effect of the crossing air flow at a lower magnitude $\frac{U_{in}}{U_{CF}}$. This suggests that fan designs with high pressure characteristics are preferable in environments where a cross flow exists. It also highlights the issues when using miniature fans under the
influence of elevated levels of cross flow.

An offsetting effect on local heat transfer was noted in the previous chapter for axial fans where the swirling outlet flow interacted with the motor support on the exit flow plane. For cooling of discrete heat sources therefore, the results for both axial and radial fans underline important considerations for positioning components which are cooled by fan induced flows. When using a radial fan arrangement as examined in this chapter with a cross flow, such discrete heat sources should be positioned approximately $90^\circ$ from the highest point of inlet flow resistance, in the direction of the fan rotation, to maximise heat dissipation. In figure 7.38, this is the location of component 3. This component, along with components 1, 2, and 4, have dimensions $5\text{mm} \times 5\text{mm}$ and are positioned $90^\circ$ apart on the base plate tangential to the fan blade tip. Figure 7.38 a) and b) illustrate the surface heat transfer distribution on the base plate for the case without cross flow and a case with $U_{CF} = 1.48\text{m/s}$ and $H_{CF} = 6.7\text{mm}$ using a Ø15mm fan at 10,000 rpm. As with all local distributions presented, the cross flow is in the direction from top to bottom of the image. The mean component heat transfer coefficient for the axisymmetric distribution without cross
flow or confinement is 136W/m²K. Component heat transfer performance is presented in figure 7.38 c) relative to the case without cross flow or confinement (NCF). A component positioned at location 3 provides the best performance for the majority of cases. The average thermal performance over all cases examined was 96% of the maximum achieved when there is no cross flow or confinement. Components 1 and 2 generally provided the worst performance and should be avoided as degradation in the heat transfer coefficient of almost 40% can be experienced at these locations.

In terms of radial heat sink design, where the overall objective is to efficiently spread the heat load from a heat source to a larger convective surface area for the fan to cool,
the asymmetric distributions discussed are not an issue once the global heat transfer levels are unaffected. The radial average of the base plate Nusselt number is presented in figure 7.39 for the Ø15mm fan at 10,000 rpm under the influence of various cross flow velocities. Results for the base plate are presented first, as it is indirectly influenced by the cross flow unlike the top plate. The main interest is to determine the indirect influence of the cross flow to the heat transfer performance within the finless heat sink. However, results for the top plate region behind the fan inlet are also discussed to provide an estimate of the trend in heat transfer under various cross flow velocities. The cases without confinement were selected to assess the influence of cross flow velocity magnitude and deviate from the negative contributions of confinement which has been previously discussed in section 7.1.3.

Beneath the fan back plate to the fan tip, the Nusselt number is independent of cross flow velocity. This is expected as the couette type flow in this region enhances heat transfer, and is only a function of fan rotational speed which is constant for the four cases presented. Immediately outside the fan tip, clear differences in Nusselt number are observed which are related to the cross flow velocity. Increasing magnitudes of $U_{CF}$ results in a reduction in the peak Nusselt number, particularly for the maximum cross flow velocity of 2.06m/s. For $0.5 < r/D < 1.25$, the introduction of a cross flow degrades the radial heat transfer distribution. The existence of a secondary vortex in the flow field results in a reduction in $Nu_D$

![Figure 7.39: Mean radial distribution of Nusselt number on the base plate for a Ø15mm fan with $H_f = 4mm$ operating at 10,000 rpm and under the influence of various cross flow velocities.](image)
for $1.25 < r/D < 1.75$ for the case without cross flow. This depression in the heat transfer trend is also evident when $U_{CF} = 0.55 \text{m/s}$ however the Nusselt number is increased. A further increase is observed for the remaining cross flow velocities examined. The secondary peak in heat transfer increases the Nusselt number above the cases with a cross flow for the remaining $r/D$ shown. In general, an increasing cross flow velocity magnitude results in a decreasing radial average in heat transfer. This is also shown in figure 7.40 for the area averaged Nusselt number and fan rotational speeds of 5000 rpm and 10,000 rpm using the Ø15mm fan. Interestingly, the radial location of the maximum $\overline{Nu}_D$ decreases with increasing cross flow levels. This is despite the extended regions of increased heat transfer shown in the local measurements of figure 7.32 over the case without cross flow. This suggests that regions of reduced heat transfer due to the local reduction in fan performance dominate the area surrounding the fan.

For $1 < r/D < 2.5$, the average reduction in $\overline{Nu}_D$ increases with increasing cross flow velocity magnitude $U_{CF}$. Operating at 5000 rpm, this reduction in area averaged heat transfer changes from 5.8% for $U_{CF} = 0.55 \text{m/s}$, to 27.3% for $U_{CF} = 2.06 \text{m/s}$. Changing the fan rotational speed to 10,000 rpm, the corresponding effect of cross flow on the heat transfer is reduced to 1.9% for $U_{CF} = 0.55 \text{m/s}$, and 17.4% for $U_{CF} = 2.06 \text{m/s}$. The standard deviation of these average values over the $r/D$ range considered are 1-1.5%. The area average in

![Figure 7.40: Area average Nusselt number on the base plate for a Ø15mm fan with $H_f = 4\text{mm}$ operating at a) 5000 rpm and b) 10,000 rpm under the influence of various cross flow velocities.](image-url)
non-dimensional heat transfer is also presented for the larger diameter fan in figure 7.41 for a rotational speed of 4000 rpm. Similarly, reductions in thermal performance are seen at the higher cross flow velocities. The reductions in $\overline{\text{Nu}}_D$ when $U_{CF} = 0.55$ m/s are insignificant over the entire $r/D$. Increasing the cross flow velocity magnitude to 2.06 m/s however decreases the area average Nusselt number by approximately 11%.

In figure 7.42, the top plate area average Nusselt number for the Ø15mm and Ø59mm fans at constant rotational speeds of 10,000 rpm and 4000 rpm are presented for various cross flow magnitudes. This surface is both directly and indirectly influenced by the crossing air flow. The top surface of the top plate is directly cooled by the crossing air stream. The lower surface of the top plate is cooled by the fan outlet flow, which is indirectly affected by the crossing air flow above the fan inlet. The results are for the measurement region behind the fan inlet, as shown in figure 4.34. Although this does not encompass the full area average around the inlet orifice of the top plate, the data of figure 7.42 provides an estimate of the heat transfer trend observed on the top plate with varying cross flow magnitude.

Unlike the base plate heat transfer distributions, the top plate Nusselt number is increased when a cross flow is introduced above the fan inlet and top plate surface. This is expected, as a crossing air flow also dissipates heat from the top surface of the top plate to increase the top plate heat transfer over the case without a cross flow, despite reducing
heat transfer performance within the finless channel. Increases in $\overline{Nu}_D$ of 35 - 100% are apparent in figure 7.42 a) by introducing cross flows of magnitude 0.55 - 2.06 m/s. Lower enhancements in $\overline{Nu}_D$ of 9 - 48% are observed for the larger Ø59mm fan in figure 7.42 b) for the same cross flow velocities. It is postulated that the lower percentage increase in $\overline{Nu}_D$ for the Ø59mm fan is due to the opposing effect of the fan inlet flow on the cross flow, previously described through figure 7.35.

The benefits and reductions in local thermal performance have been discussed in this section. Analysis of the global thermal performance within a finless heat sink suggests that a crossing air flow above the fan inlet will indirectly result in a decrease of the cooling solution thermal performance. However, it is estimated that the cross flow benefits heat transfer performance on the top plate when allowed to directly cool the top surface. In certain circumferential locations, the fan is forced to operate near the rotating stall point, considerably reducing thermal performance. It is also anticipated that the full benefits of the contributing cross flow are not realised due to the inlet flow distortion that arises over the entire fan inlet. An off-axis, swirling inflow results from the rotating fan and this may approach the blades at an unfavourable angle of incidence. The local contributions of the cross flow to fan performance, and consequently thermal performance within the finless channel, are therefore outweighed by the degrading effects.

Figure 7.42: Area average Nusselt number on the top plate for a) Ø15mm fan with $H_f = 4$ mm operating at 10,000 rpm and b) Ø59mm fan with $H_f = 4$ mm operating at 4000 rpm under the influence of various cross flow velocities.
CHAPTER 7. HEAT TRANSFER WITH RADIAL FANS

7.3 Closure

This chapter examined the velocity fields and local surface heat transfer distributions for radial fan and radial finless heat sink cooling solutions. Fan performance was characterised for miniature radial fans highlighting the scaling effects which can exist at low blade Reynolds numbers. The limitations of fan profile scaling on both the fan performance and the local and average heat transfer distributions within the finless heat sink were also presented. It was determined that radial fan and radial finless heat sink designs should be limited to $a_r = 0.3$ to avoid substantial degradation in thermal performance. Local and area averaged heat transfer distributions were acquired for a constant fan profile with a number of scaled diameters. Similar fluid structures created by each radial fan diameter with a geometrically scaled design were found to occur within the finless channel. This resulted in similarities of the surface heat transfer distribution for all fans which scaled towards a turbulent flow regime due to the fluid unsteadiness generated by rotating fan assemblies. Unsteady fluid structures such as corotating and counter rotating vortex pairs were found to interrupt the shear flow along the base surface and promote turbulent diffusion which increased heat transfer performance.

A comparison between radial heat transfer coefficient distributions for miniature axial and radial fan designs investigated in this thesis was considered. It was determined that the radial fan designs in this chapter provide the greatest level of heat transfer for the power and space required.

Finally, a study on the influence of practical operating conditions was chosen for radial fans at either end of the diameter scale considered. Varying confinement heights and cross flow levels above the fan inlet were found to indirectly result in a reduction in heat transfer performance both locally and when integrated over an area centred on the fan rotational axis. The reduction in thermal performance was shown to increase for an increasing cross flow velocity magnitude, in some cases resulting in a degradation in global heat transfer of 30% for the smallest radial fan arrangement. Despite the unfavourable conditions produced within the finless channel, a cross flow directly cools the top surface of the top plate, leading to increases in thermal performance on the top plate over the case without cross flow. A substantial degradation which resulted in an asymmetric local heat transfer distribution
was observed when the fan was forced to operate in a local stall region. Recommendations for component positioning were also highlighted for the example of a 5mm × 5mm area representative of an electronic component to maximise heat transfer performance when under the influence of a cross flow. Any benefits of increased flow rate at certain circumferential locations of the outlet flow on heat transfer were found to be outweighed by the stalling and distorted inflow effects that a cross flow produces.
Chapter 8

Conclusions and Recommendations

8.1 Conclusions

Experimental investigations into the performance and enhancement of miniature and low profile heat sinks along with direct component cooling were completed. Local velocity field and heat transfer measurement techniques were used to assess the fluidic mechanisms produced by fan induced flows which are fundamental to cooling solution performance optimisation and prediction. The main findings of this thesis are compiled below and can assist thermal engineers in the design of fan induced, forced convection cooling solutions for portable and space constrained applications.

Miniature finned and finless heat sink geometries:

- A reduction in thermal resistance can be obtained for miniature fan - heat sink designs by accounting for fan exit angles, and aligning the exit flow with the heat sink channels using a diffuser between the fan exit and heat sink inlet. In the analysis of one of the smallest commercially available radial blowers, thermal resistance has been reduced by up to 23% for a finned heat sink and 15% for a finless heat sink over the typical arrangement of fan outlet and heat sink inlet positioned adjacently. In order to achieve these reductions the footprint area has been increased by 16% and 6%, respectively.
• Benefits in heat transfer performance and power usage can be achieved by implementing a finless design as opposed to the conventional finned designs in space restricted environments. This was particularly evident in the experimental studies on heat sink profile scaling, as forced convection heat transfer enhancements of up to 55% were obtained using the finless design when heat sink profile height was decreased below 4mm.

• Higher heat transfer rates are achieved with a finless cooling solution than predicted using theory for single fluid heat exchangers. This can be attributed to the flow patterns created with the radial flow fan and the larger flow area finless designs, highlighting the necessity to consider both fan and heat sink collectively in the design process rather than separately, as is traditionally accepted.

• Finned and finless geometries are optimised when $L_{Dh}^* \approx 0.1$. This criterion for optimisation physically defines the point where the thermal boundary layers merge at the heat sink exit and also agrees with that found in the literature for optimisation of conventional heat exchanger designs. The current work presents an empirically determined cross over criterion in design choice. This can be utilised by thermal engineers to confidently design finless geometries which will outperform conventional finned designs for use in low profile, space constrained environments.

Component placement:

• Strategic positioning of components relative to radial and axial fan flows can provide significant enhancements in average component heat transfer. In the experimental cases presented, these enhancements were up to threefold for relatively small positional changes. Non-uniform and complex exit flows from rotating fans must be assessed to optimise the position of discrete heat sources for maximum heat dissipation. For radial fans, positioning heat sources adjacent to the fan blades resulted in maximum heat dissipation levels. Due to the complex fan exit flows produced, local regions of fluid impingement existed and also promoted thermal performance. For axial fans, discrete heat sources should be located beneath the widest motor support
when motor supports are positioned on the exit flow plane and $H/D \leq 0.309$. The results presented in the current work on component placement, relative to radial and axial fan designs, can be used as a guideline for thermal engineers.

Local surface heat transfer measurement technique:

- The uncertainties associated with a combined infrared thermography and heated-thin-foil technique, for application to low heat transfer coefficient distributions $h_{fc} < 10^2 \text{W/m}^2\text{K}$, were clarified experimentally. Using an energy balance method, and measuring the true foil and paint thicknesses, heat transfer mechanisms can be accurately accounted for. These parameters should be assessed when implementing this local measurement technique for low convective heat transfer coefficients and complex heat transfer gradients to ensure accuracy. A useful procedure has been outlined that only requires a simple one-dimensional analysis of the foil surface temperature to estimate if tangential conduction can be neglected, and also if the measurement of both foil and paint thicknesses is required.

Axial fans:

- Local heat transfer coefficients on a flat surface due to impinging axial fan flows were found to be non-uniform. Fan motor supports have been identified as the main contribution to the variation in heat transfer coefficient over the flat surface. Local regions of increased heat transfer are a function of fan distance from the impingement surface, and independent of fan rotational speed above a critical Reynolds number. At $Re_B < 2000$, the aerodynamic performance of the fan suffers from an adverse flow phenomena that reduces fan performance and increases swirl angle.

- Fluctuations in surface heat transfer coefficient are directly linked to the unsteady fluid structures generated by axial fan blade-motor support interaction. The radial distribution of non-dimensional heat transfer was presented for miniature and large scale designs indicating a common scaling relationship of $Nu_d \sim Re_d^{0.6}$ for $60 < Re_d < 3000$, highlighting the significance of the unsteady, swirling flow on surface heat transfer augmentation.
• Positioning motor supports on the inlet flow plane of axial fan designs was found to be the most beneficial practical arrangement for surface heat transfer enhancement and uniformity. This is particularly important in the design of miniature axial fans, as motor supports accommodate a greater percentage of the fan flow area than for larger diameter macro scale designs. The ideal case without any motor supports was found to provide approximately 20% increase in radial heat transfer over the other arrangements. Therefore, miniature axial fans should be designed with inlet mounted motor supports which have minimal dimensions without compromising mechanical strength.

Radial fans:

• Miniature radial fans should be designed with an aspect ratio less than 0.3 to avoid reductions in thermal performance. This has been attributed to a reduction in flow coefficient, as no further increase in flow rate is achieved above this aspect ratio. Consequently, the mean velocity within the finless channel reduces as the plate spacing increases.

• A radial fan outlet flow, confined between two parallel plates, produces a high velocity shearing flow along the base plate which results in a shear layer forming between the low velocity fluid in the finless channel and the high velocity fan exit flow. It is postulated that this exit flow profile occurs due to inertia forces, as the fan directs the axial inlet flow to a radial outlet flow. As aspect ratio decreases and flow rates increase, this effect is reduced and the flow is more uniformly distributed over the entire blade height.

• Primary and secondary counter rotating toroidal vortices in the time-averaged flow field result in promoting heat transfer on the top plate and also over an annular region on the base plate of a radial finless cooling solution design. As flow rate through the finless channel increases, the secondary vortex is overcome by the mean flow and the resultant annular region of increased heat transfer on the base plate is no longer evident.
• Instantaneous velocity field measurements suggest that turbulence within the finless channel results from the interaction of counter rotating and corotating vortices with the high shear flow. The interruption of the fan exit flow by these unsteady fluidic mechanisms disrupts boundary layer growth on both finless plates, promoting heat transfer. The enhancement due to the unsteady flow field is reflected in the scaling relationship with Reynolds number as $Nu_D \sim Re_D^{0.65}$ for $645 < Re_D < 34560$.

• Miniature radial fans are superior to miniature axial fans for thermal management in portable and low profile applications based on cooling solution space and power requirements. A radial fan design was shown to provide an increase in thermal performance of 25% over an axial fan design of equal footprint area and fan power. The total profile height required by the radial fan was also only one third of the overall height of the axial fan arrangement.

• The introduction of a crossing air flow above a radial fan inlet indirectly reduces both the local and global thermal performance of the finless cooling solution. A swirling inlet flow is produced, and local variations in static pressure can result in local stall. It is hypothesised that the combined effect of a swirling inflow and whirling outlet flow offsets surface heat transfer from an axisymmetric to asymmetric distribution, which is dependent on the fan rotational direction. Optimum positioning of components relative to a radial fan can maintain the average component heat transfer coefficient at a similar level to a case without any cross flow.

• An increase in cross flow velocity magnitude, indirectly decreases thermal performance within the finless channel. For a Ø15mm radial fan, the reduction in area averaged thermal performance was 6% over $1 < r/D < 2.5$ at a fan rotational speed of 5,000 rpm and cross flow velocity of $U_{CF} \approx 0.55\text{m/s}$. Increasing the cross flow velocity magnitude to $U_{CF} \approx 2\text{m/s}$ resulted in a reduction of almost 30%.


8.2 Recommendations

The experimental observations in this thesis have provided an insight into the fluid dynamics and thermal enhancements when implementing forced convection cooling using rotating fans at miniature and low profile scales. A number of recommendations for future research have emerged from the findings and are listed below.

- The infrared thermography and heated-thin-foil technique for steady state analyses was developed and assessed for non-uniform fan flows. An experimental assessment of the unsteady state heated-thin-foil technique would determine if the energy balance method presented is suitable for resolving the instantaneous heat transfer coefficient and the limited frequency of the fluctuations. In the steady state analysis, the contribution of the paint layer was relatively minor due to a low thermal conductivity. As the transient measurements are affected by the volumetric heat capacity of the material, the paint layer has a major influence as it is approximately 1.6 times that of the stainless steel foil. This also highlights the importance of developing isotropic thin-foil materials which are electrically resistive, have a high surface emissivity without the use of paint coatings, and low thermal conductivity and volumetric heat capacity. This material would therefore reduce the influence of spatial and temporal conduction when resolving local heat transfer coefficient measurements.

- Surface heat transfer measurements due to axial fan impingement have indicated regions where low heat transfer exist beneath the hub, and also regions where high heat transfer occur outside the fan housing. An investigation into heat sink designs which can promote heat transfer beneath the fan hub and prevent the expanding exit flow from by-passing the heat conducting fins would be beneficial for electronics cooling applications. Low profile heat sink designs also require further research as typical axial fan - finned heat sink cooling solutions have $H/D \approx 0.5$. With knowledge of the optimum positioning of components highlighted in this thesis using axial fan flows, it would be beneficial to analyse the significance on heat transfer of raised components in these positions, as heat transfer due to local fluid impingement was briefly shown to benefit from a raised component orientation.
• Simultaneous velocity field and local heat transfer measurements using high speed PIV and infrared thermography systems would provide further insight into the effect of vortex interaction within finless channels on local heat transfer. Using a stereo PIV technique, which acquires three components of velocity, this type of analysis could determine the effect of three-dimensional fluid structures on surface heat transfer.

• A radially discharging fan design under the influence of cross flow resulted in an asymmetric local heat transfer distribution which was offset in the blade rotational direction. Experimentation to determine the effects of a cross flow on voluted radial fan designs would also be beneficial. This could determine the orientation that adjacent fan - heat sink designs should be, relative to a cross flow, to reduce the degradation of heat transfer.

• Phase-locked PIV measurements of the fan blade passage at different circumferential locations around the fan could provide conclusive evidence to the cause of the offset in heat transfer when a cross flow is introduced above a radial fan inlet. This would illustrate the angle of the distorted inlet velocity relative to the blade leading edge, and the exit flow towards the blade tip.

• The indirect influence of a crossing air flow on heat transfer in a radial finless heat sink was examined. An investigation into the effect of a crossing air flow between the finless plates, as well as above the fan inlet, would highlight the direct influence of a cross flow on fan exit flow and thermal performance.
References


management using phase change materials with embedded graphite nanofibers for systems with high power requirements, 11th Intersociety Conference on Thermal and Thermomechanical Phenomena in Electronic Systems (ITherm), Orlando, FL, May 28-31, pp. 561–566.


REFERENCES


REFERENCES


REFERENCES


Appendix A

Publications

Journals


**Conferences**


Appendix B

Experimental Appendix

B.1  Calibrations

B.1.1  Temperature recording instrumentation

A Lauda E100 RE104 water bath, pre-calibrated by the manufacturer to 0.1°C, was chosen for calibrating the temperature measurement instrumentation. This instrumentation consisted of diameter 0.25mm K-Type thermocouples, NI 9211 thermocouple input module with USB-9161 carrier, and a Compaq EVO N410c personal computer for data acquisition. The instrumentation was calibrated collectively to reduce the overall systematic error in the temperature measurement.

In figure B.1, sample calibration data is presented for seven measurements and a range

![Figure B.1: Calibration data for a number of temperature measurements](image-url)
of temperatures spanning approximately 15 - 95°C. In the case presented, the systematic error is almost the same for all seven measurements and has a linear relationship with temperature. The maximum error of all temperature measurements in figure B.1 is 1.1°C. Using this calibration procedure it was possible to reduce the uncertainty in the temperature measurements to 0.1°C.

B.1.2 Fan characteristic rig

In figure B.2, the calibration charts for the various orifice diameters are shown.

![Orifice calibration charts](image)

**Figure B.2: Orifice calibration charts**

B.1.3 Infrared camera (Temperature)

Infrared thermography utilises the relationship between electromagnetic radiation and temperature to provide a visual image of the thermal energy emitted from an object. As this method of detection is non-intrusive and fully two-dimensional, it can serve various applications, many of which have been outlined by Meola and Carlomagno (2004). An infrared camera, including optics and filters, and a personal computer for camera control and/or image post processing, provide the basis of a thermographic system. The camera contains infrared detectors, which absorb electromagnetic radiation emitted by the object and
convert it into an electrical signal. The electromagnetic radiation detected by the camera however, is only a portion of the total electromagnetic radiation emitted by the object. Infrared cameras contain detectors which only absorb electromagnetic radiation over certain wavelengths. The wavelength intervals within the infrared spectral band are commonly referred to as: near infrared (0.75 - 3\(\mu\)m), middle infrared (3 - 6\(\mu\)m), far infrared (6 - 15\(\mu\)m), and extreme infrared (15 - 1000\(\mu\)m). An Indigo Merlin Mid camera, with an Indium Antimonide (InSb) Focal Plane Array (FPA), was used for capturing local heat transfer trends over the heated-thin-foil. The FPA consists of a 320\(\times\)256 staring array of detectors which are sensitive from 1.0\(\mu\)m to 5.4\(\mu\)m. However, a cold filter is incorporated in the standard camera configuration which limits the camera’s spectral response to 3.0\(\mu\)m – 5.0\(\mu\)m. A Non-Uniformity Correction (NUC) procedure, carried out by the manufacturer, compensated for the variation in the individual response of the detectors to thermal energy. In order to provide enhanced temperature contrast, the dynamic range of the 12-bit camera output is calibrated over short temperature intervals. The camera has been pre-calibrated over four intervals, giving an overall temperature range of approximately –20\(^\circ\)C to 350\(^\circ\)C. The manufacturer calibration certification is included in the following section.

The emissive power of an ideal blackbody observed by the camera can be calculated by integrating Planck’s law over the camera wavelength (Holman, 2002)

\[
E_b(3-5\mu m) = \int_3^5 \frac{C_1}{\lambda^5 \left(e^{C_2/\lambda T} - 1\right)} d\lambda \quad (B.1)
\]

where \(C_1\) and \(C_2\) are the first and second radiation constants, \(\lambda\) is the wavelength of the blackbody radiation, and \(T\) is the absolute temperature of the blackbody.

Although real objects can approach the case of a blackbody, most have surface characteristics that result in their radiance emittance being a fraction of the radiance emitted by a blackbody at the same temperature and wavelength. The radiance emittance of a real object can therefore be described as a function of the blackbody emittance through the introduction of the spectral emissivity coefficient, \(\varepsilon\).

\[
E_{obj}(\lambda, T) = \varepsilon(\lambda).E_b(\lambda, T) \quad (B.2)
\]
As shown in Eq. (B.2), the emissivity of a real object can also be dependent on wavelength. Figure B.3 presents the emissive power over a portion of the infrared wavelength band for the example of a) a blackbody, b) emissivity independent of wavelength or greybody, and c) a real surface, for an object temperature of 380K. By painting the heated-thin-foil surface matt black, a high emissivity finish was achieved, which was measured using ISO 18434-1 (2008) standard.

Considering infrared detectors only capture the emissive power of an object over a limited bandwidth, it is possible to conclude that measurements acquired must follow a similar form as Planck’s law (Eq. B.1). Infrared detectors, however, detect energy from the object of interest along with surrounding contributions from the atmosphere and nearby sources of radiation. These contributions, or offsets, must therefore be accounted for to resolve the object radiance emittance and the object temperature. ThermaCam Researcher Pro 2.8 was the software used to estimate these contributions to the total emissive power detected by the infrared camera. The basic thermographic measurement situation is shown in figure B.4. The measurement formula used by the software is provided in Eq. (B.3)

\[ E_{\text{tot}} = \varepsilon \tau E_{\text{obj}} + (1 - \varepsilon) \tau E_{\text{refl}} + (1 - \tau) E_{\text{atm}} \]  

\[ (B.3) \]

where \( E_{\text{tot}} \propto U_{\text{tot}} \), the camera output signal.

The camera user must input the relevant parameters of object emissivity, \( \varepsilon \), atmospheric temperature, reflected temperature, object distance, and relative humidity to solve for the
object signal, \( U_{obj} \). Object emissivity and reflected temperature were measured as per ISO 18434-1 (2008). A K-type thermocouple was used to measure atmospheric temperature, and a Precision Gold N18FR hygrometer monitored the relative humidity.

As previously mentioned, the pre-calibrated camera measurement range is divided into four intervals over approximately -20°C to 350°C. These intervals, defined NUC0 - NUC3, have specific calibration parameters. Operating in NUC1 provided an approximate temperature measurement range of 20°C to 115°C which was sufficient for the heated-thin-foil measurements carried out. For practical application to cooling of portable electronics, the heated-thin-foil was not required to exceed the upper limit of this measurement range. The camera manufacturer specifies a temperature accuracy of 2°C or 2% of the temperature measurement. To achieve an improvement on this accuracy, a separate calibration procedure was required. The measurement formula shown in Eq. (B.3) is also a general method for calculating object temperature based on an idealised experimental arrangement which can be difficult to replicate in practice. It is therefore necessary to perform an in situ calibration of the infrared camera to adequately account for radiation offsets exclusive to the experimental arrangement and reduce experimental errors.

A similar in situ calibration approach as described by Schulz (2000) was implemented. The infrared camera uses a modified Planck formula to calculate temperature from the camera signal. This formula is shown in Eq. (B.4)
where \( R, B, \) and \( F \) are both camera (including lens and filter) and NUC1 dependent constants that take into account Planck’s constants in Eq. (B.1).

The experiment was arranged as described in section 4.1.1. Three K-type thermocouples were used to measure the surface temperature of the heated-thin-foil in discrete positions where large temperature gradients were avoided and spanning the image field of view (FOV). The user-defined parameters mentioned previously were specified in the software to account for the general offset radiation contribution. Once the heated-thin-foil reached steady state, the camera output signal was recorded. The object signal was then averaged over a 1mm\(\times\)1mm area which was centred at each thermocouple location. The relationship between the theoretical object emissive power (solving Eq. (B.1) with experimental temperature measurements) and the camera output signal, is shown in figure B.5. This confirms the linear proportionality of the camera output signal over the entire NUC1 measurement range considered.

Using the Levenberg-Marquardt method introduced by Levenberg (1944) and developed further by Marquardt (1963) for non-linear least squares problems, a non-linear least squares fit was achieved between the thermocouple reference temperature and \( T_{obj} \). For each measurement point, the difference between the reference temperature and the infrared

![Graph](insert_graph.png)

Figure B.5: The relationship between emissive power and camera output signal.
camera temperature (solving Eq. (B.4)) is

\[ D_i = T_{i(\text{ref})} - \frac{B}{\ln\left(\frac{R}{U_i} + F\right)} \]  

(B.5)

The variances were then minimised such that

\[ D_i^2 = T_{i(\text{ref})}^2 - \frac{BT_{i(\text{ref})}}{\ln\left(\frac{R}{U_i} + F\right)} + \frac{B^2}{\left[\ln\left(\frac{R}{U_i} + F\right)\right]^2} \Rightarrow 0 \]  

(B.6)

The modified constants \( R, B, \) and \( F \) were then used in Eq. (B.4) to replace the original constants and to provide an accurate temperature field for further data analysis. The calibration curve for the experimental configuration carried out is presented in figure 4.5. A comparison of the original infrared camera error and the reduction in error by implementing this \textit{in situ} calibration is also presented in figure 4.6. Over the entire NUC1 measurement range, temperature uncertainty was reduced from 2.8K to 0.2K.

### B.1.4 Infrared camera (NUC)

In section B.1.3, an \textit{in situ} camera calibration method was presented to refine the accuracy of the local temperature measurements using an infrared camera. Figure B.6 presents the certificate of calibration by the manufacturer for Non-Uniformity Correction.
Figure B.6: Calibration certificate for FLIR Indigo Merlin Mid infrared camera.
B.2 Assessment of a heated-thin-foil top plate

The experimental set-up and procedure for the investigation of the local heat transfer distribution on the base and top plates of a finless heat sink is presented in section 4.1.4. The heated-thin-foil for the top plate experiments was positioned tangential to the fan inlet orifice as shown in figure 4.34. This arrangement was chosen due to the non-uniform input heat flux \( q''_{\text{gen}} \), generated when a top plate with an inlet orifice is Joule heated. The discontinuity of the thin-foil results in substantial temperature variations due to the non-uniform \( q''_{\text{gen}} \) along the foil. This is indicated in figure B.7. For the example shown, variations in surface temperature of over 20K exist around the orifice circumference.

![Temperature distribution of a heated-thin-foil with a Ø15mm orifice. Contour level: 0.6K.](image)

Figure B.7: Temperature distribution of a heated-thin-foil with a Ø15mm orifice. Contour level: 0.6K.
B.3 Fan motor and shaft support housing

Figure B.8 presents the fan motor and extended shaft arrangement which was part of the IR experimental rig used for investigating local heat transfer distributions for radial fans. This was designed primarily to avoid obstructing the IR camera field of view. It was also implemented for the analysis of axial fans with modified motor support arrangements.
B.4 Sample uncertainty calculation

A sample uncertainty calculation of the measured thermal performance of a heat sink considered in this thesis is provided below. Using Eq. (3.12) from section 3.3, and the uncertainty in the measured variables, the uncertainty in the data presented for forced convection thermal resistance ($R_{fc}$) can be calculated. The forced convection thermal resistance is calculated using the total thermal resistance and resistance due to secondary losses as discussed in section 3.1.1. The total thermal resistance is

$$R_{tot} = \frac{(T_w - T_i)}{VI}$$  \hspace{1cm} (B.7)

The estimated uncertainty in the measured variables ($\delta$) of this equation are:

- $T_w = 339.08K$, $\delta T_w = 0.1K$
- $T_i = 293.75K$, $\delta T_i = 0.1K$
- $V = 5.18V$, $\delta V = 0.01V$
- $I = 0.592A$, $\delta I = 0.01A$

$$\delta R_{tot} = \left\{ \left[ (R_{tot}(T_w+\delta T_w) - R_{tot}(T_w))^2 + (R_{tot}(T_i+\delta T_i) - R_{tot}(T_i))^2 \right] + \left[ (R_{tot}(V+\delta V) - R_{tot}(V))^2 + (R_{tot}(I+\delta I) - R_{tot}(I))^2 \right] \right\}^{1/2}$$

$$\delta R_{tot} = (14.814 - 14.782)^2 + (14.749 - 14.782)^2 + (14.753 - 14.782)^2 + (14.536 - 14.782)^2 \left\{ \right\}^{1/2}$$

$$\delta R_{tot} = 0.251K/W$$

Similarly, the estimated uncertainty in the measured variables for $R_{losses}$ are:

- $T_w = 339.69K$, $\delta T_w = 0.1K$
- $T_i = 294.77K$, $\delta T_i = 0.1K$
\[ V = 4.00 \text{V} \quad \delta V = 0.01 \text{V} \]
\[ I = 0.458 \text{A} \quad \delta I = 0.01 \text{A} \]

which indicates an uncertainty in the secondary losses measurement of \( \delta R_{\text{Losses}} = 0.533 \text{K/W} \).

Using the calculated uncertainty in both the total and secondary losses thermal resistances, the uncertainty in the forced convection thermal resistance can be determined.

\[ R_{\text{tot}} = 14.782 \text{K/W} \quad \delta R_{\text{tot}} = 0.251 \text{K/W} \]
\[ R_{\text{Losses}} = 24.516 \text{K/W} \quad \delta R_{\text{Losses}} = 0.533 \text{K/W} \]

\[ R_{f c} = \frac{R_{\text{tot}} R_{\text{Losses}}}{R_{\text{Losses}} - R_{\text{tot}}} \quad (B.8) \]

\[ \delta R_{f c} = \left\{ \left( R_{f c} (R_{\text{tot}} + \delta R_{\text{tot}}) - R_{f c} (R_{\text{tot}}) \right)^2 + \left( R_{f c} (R_{\text{Losses}} + \delta R_{\text{Losses}}) - R_{f c} (R_{\text{Losses}}) \right)^2 \right\}^{1/2} \]

\[ \delta R_{f c} = \left\{ (38.866 - 37.229)^2 + (36.064 - 37.229)^2 \right\}^{1/2} \]

\[ \delta R_{f c} = 2.001 \text{K/W} \]

The relative uncertainty for the measurement of \( R_{f c} \) can be calculated using Eq. (3.13) from section 3.3.

\[ u_{R_{f c}} = \frac{\delta R_{f c}}{R_{f c}} = \frac{2.001}{37.229} = 5.37\% \]