Transactions on Engineering Sciences vol 20, © 1998 WIT Press, www.witpress.com, ISSN 1743-3533

Characteristics of heat sink flow bypass for thermal modeling

P. Gauché, C. Coetzer, & J.A. Visser
Department of Mechanical and Aeronautical Engineering, University of Pretoria, Pretoria, 0002, South Africa
E-Mail: paul.gauche@eng.up.ac.za

Abstract

Extruded aluminium heat sinks are used ubiquitously in the electronics industry to cool chips, modules and packages because of their simplicity and effectiveness. Due to rapidly increasing chip power dissipation and the need to minimize weight, size, cost and time-to-market, there is a great need to accurately predict the heat transfer in heat sink assemblies.

This study involves an investigation of the airflow for forced convection heat sinks. These high performance heat sinks offer a relatively large flow resistance causing flow bypass and leakage from the heat sink. This effectively lowers the fin flow velocity and therefore the performance of the heat sink. A parametric study using Computational Fluid Dynamics (CFD) was conducted to determine the characteristics of the flow through and around a typical heat sink. Parameters such as fin height, fin gap, bypass width heat sink length and were investigated. Analytical expressions were used to develop a compact model of the flow bypass and pressure drop to be incorporated into a computational heat transfer computer code that can assist the designer in achieving all the minimization criteria. The model was found to be accurate compared with published experimental results.

1 Introduction

Increasing power dissipation as well as increasing chip densities in the rapidly evolving electronics industry is causing an ever increasing need for tools and methods for optimal design. This is further exacerbated by the competitive need to produce smaller, more powerful equipment with short product life cycles. Advances in the performance of computers and in computational methods have led to the availability of various useful commercial Computational Fluid Dynamics (CFD) software packages. This software solves the discretised form of the conservation equations of mass, momentum and energy to provide a solution of the flow and heat fields in the domain of interest. This method of using the fundamental flow physics is currently the most complete way to determine the solution in such problems. It does however have the drawback of being computationally expensive, requiring relatively powerful equipment and a fair amount of patience.

The electronics cooling community has identified a need for lumped-parameter models, which have been termed "compact models" to assist in the design of electronic products. These models could assist the designer in system level analysis where the use of CFD is prohibitive due to the requirement of large computational models. A European funded project termed *Delphi* (Development of Libraries and Physical Models for an Integrated Design Environment) was established to determine such models and have delivered good results [1].

A subject enjoying much attention in the electronics cooling community is the use of passive forced convection heat spreaders usually called heat sinks. Due to the relative simplicity and cost involved, these devices have become common practice in electronics assemblies ranging from personal computers (PC's) to Telecommunication power electronics. Despite the advantages of these devices, there are serious consequences involved in the application of heat sinks. The necessary size of a heat sink can prevent the miniaturization of a product substantially and the lack of understanding of the flow physics can lead to inadequate designs causing failure. An important reason for the above-mentioned inadequacies is a phenomenon termed flow-bypass. The location or placement of the heat sink is as important as the selection of a heat sink for the following reason. A heat sink requires two principle properties: a large surface area to dissipate heat and the ability to accept a high flow rate *through* the heat sink. High performance heat sinks unfortunately tend to have a high flow resistance causing a large pressure drop across the system. If the heat sink is in a ducted system with tip and lateral clearances, a large portion of the flow tends to bypass the heat sink effectively lowering the flow rate through the heat sink and therefore the performance.

This focus of this work is based on the mechanisms of flow bypass with the aim of developing a compact model to simulate the flow through and around heat sinks without having to make use of computationally expensive CFD software. When coupled with models for the heat transfer, accurate results should be obtainable. A parametric study using

Transactions on Engineering Sciences vol 20, © 1998 WIT Press, www.witpress.com, ISSN 1743-3533

CFD was conducted to determine the flow mechanisms and construct a flow model for this purpose.

2 Literature on Heat Sinks and Bypass

The study of the effects of heat sinks has been ongoing for about two decades and the number of publications has increased dramatically over the last number of years. Early on, van der Pol and Tierny [2] investigated the heat transfer of parallel plate heat sinks placed vertically for natural convection. Their analytical model has been used with success and has been referenced numerously since then. Heat sink bypass articles have become increasingly prominent in journals and proceedings in the last five years. Lee [3] developed an analytical model to optimise longitudinal finned heat sinks in channeled flow with bypass. Kim and Lee [4] conducted a survey on the measurement and characterization of heat sinks with bypass indicating that there is no industry standard for testing procedures or definitions. Belady [5] reported the same problem and announced the formation of an ASME adhoc subcommittee to establish standards for heat sink characteristics in forced convection.

In order to characterize a heat sink, the so-called thermal resistance needs to be defined. This can be expressed as:

$$R_{HS} = \frac{\Delta T}{Q} = \frac{T_{HS} - T_\infty}{Q} \tag{1}$$

For a heat sink, this is the difference in temperature from the *maximum* heat sink temperature (T_{HS}) to ambient (T_∞) divided by the heat dissipated (Q). It can be noted that this value is the only value necessary in the design process, hence its definition. It is also necessary to define the pressure drop characteristic: this is the drop in pressure before and after the heat sink. The determination of the thermal resistance has traditionally been based on experiments conducted in wind tunnels.

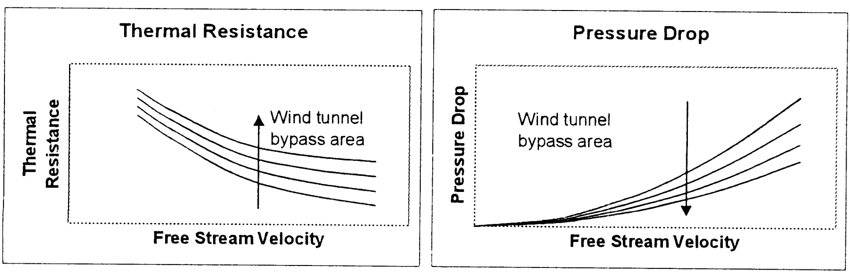

Figure 1. Effects of Bypass in Wind Tunnels

The results for both thermal resistance and pressure drop vary for a given heat sink and device assembly in different wind tunnels because of the different bypass clearances in different tunnels. Part of the problem is that the results are based on the free stream velocity and not the correct fin flow velocity. The effect of bypass can be illustrated in figure 1. These problems can be avoided by either specifying that the characterization take place with no bypass, as proposed by the committee, or by determining the actual fin flow velocity. The former is a practical solution whilst the later adds more complexity as the relevant velocity must either be measured or calculated.

Obinelo [6] makes use of a commercial CFD program to investigate and characterize the thermal and flow phenomenon for longitudinal fin heat sinks in forced convection. A similar model was developed by Butterbaugh and Kang [7], who used analytical expressions in a resistance network. Their experimental results showed good correlation with the model for a range of bypass conditions.

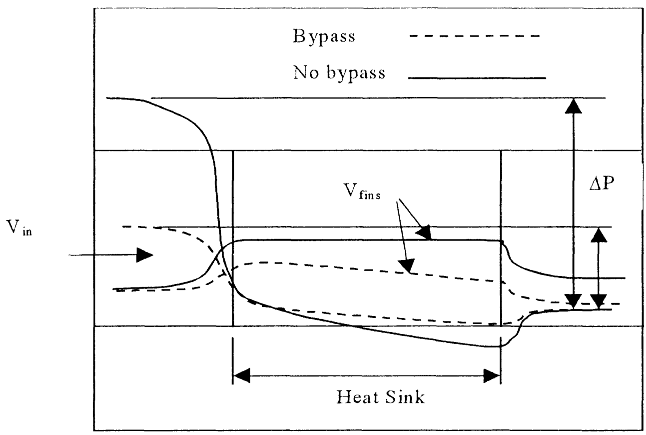

Figure 2. Pressure and Velocity Through Heat Sinks

The pressure and velocity profiles through a heat sink can be schematically represented by figure 2 for a heat sink without and with bypass. For the case with no bypass, the following observations can be made: The pressure drops rapidly at the entrance. This is attributed to the irreversable losses caused by two recirculation areas just before and after the contraction of the heat sink. A further loss between the fins is due to the frictional losses associated with the boundary layer, a steeper pressure loss gradient initially is due to the developing nature of the flow. At the exit, a partial static pressure recovery occurs as the velocity drops. The fin flow velocity is constant throught the finned region as there is no

Transactions on Engineering Sciences vol 20, © 1998 WIT Press, www.witpress.com, ISSN 1743-3533

bypass. In the bypass case, the total static pressure drop is much smaller with the same system flow rate. This is due to the bypass region having little flow impedance. To satisfy the pressure balance, the flow velocity is higher in the bypass region causing a low fin flow velocity. The open fins also cause fin leakage causing the velocity to drop further throught the heat sink.

3 Heat Transfer and Flow Bypass Modeling

The approach followed in this study to solve the heat transfer in forced convection heat sinks with discrete powered devices and including flow bypass is summarized here.

As mentioned in the introduction, a method for solving the heat transfer in heat sink assemblies without the use of CFD is the objective. To be accurate and reliable, this method needs to model the flow in and around the heat sink accurately and then needs to model the heat transfer coefficients based on the turbulence levels and velocity fields.

The heat sink itself is modeled numerically with a curvilinear form of the discretised diffusion equation. In Cartesian co-ordinates the diffusion equation is:

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x}\left(k \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(k \frac{\partial T}{\partial y}\right) + \frac{\partial}{\partial z}\left(k \frac{\partial T}{\partial z}\right) + S \tag{2}$$

The reason for using a numerical simulation is that the temperature variation through the heat sink can be accurately determined with the aim of calculating the maximum temperature (T_{HS}) for the definition of the thermal resistance (R_{HS}). This method prevents any ambiguity regarding the size and location of a device and hence any confusion regarding the formal definition of what the thermal performance of the heat sink actually is. On this note, it is worth pointing out that technically, the thermal resistance cannot be specified for a heat sink alone. The size, location and number of devices have an effect on the thermal resistance because of the finite thermal conductivity of the heat sink material. This method therefore gives an *application dependent* heat sink thermal resistance.

The boundary conditions are specified using compact analytical and empirical models. Firstly the velocity field needs to be determined, then the heat transfer coefficients are calculated to be used on the numerical boundary.

An analytical model to determine the flow velocities proposed by Butterbaugh and Kang [7] was adapted for the purpose of this study. For a given approach velocity and known duct dimensions, the airflow rate, Q

is used to determine the flow velocity in the finned region by applying pressure balance and mass conservation conditions. To determine the fin flow velocity, the flow rate in the heat sink fin channels needs to be determined. This is done by considering each flow path in the duct containing the heat sink obstruction and then determining the associated pressure losses in the system using a flow network set-up. This network identifies the flow rates (Q), velocities (V) and pressure drops (ΔP) in each section of the flow path. Subscripts identify the flow path (h = heat sink, t = top, l = lateral) and the pressure drop mechanism (i = inlet contraction, f = frictional, e = exit expansion). A pressure rise occurs in the stagnation region just ahead of the heat sink and at the exit of the heat sink. These are denoted by subscripts s and b respectively. These components are added to the bypass regions as losses and not as gains in the heat sink flow path so that the pressure drop directly across the heat sink can be calculated.

Each flow path has to conform to a pressure drop balance and mass conservation laws so that

$$\Delta P_h = \Delta P_t = \Delta P_l \tag{3}$$

and

$$Q_h + Q_t + Q_l = Q \tag{4}$$

This is achieved by estimating the initial flow rates and then calculating the sum of each individual pressure drop mechanism in each flow path. The pressure drop in each flow path is then used to redefine the flow rates in each path and the process is repeated iteratively until both equations are satisfied.

The inlet contraction losses (ΔP_i) and exit expansion losses (ΔP_e) are adapted from Kays [8] and are represented as

$$\Delta P_i = K_i \times \tfrac{1}{2}\rho V_i^2 \tag{5}$$

and

$$\Delta P_e = K_e \times \tfrac{1}{2}\rho V_e^2 \tag{6}$$

where

$$K_i = -0.41\alpha^2 + 0.02\alpha + 0.4 \tag{7}$$

$$K_e = 1.0\alpha^2 - 2.0\alpha + 1.0 \tag{8}$$

$$K_i = -0.41\alpha^2 + 0.02\alpha + 0.8 \tag{9}$$

$$K_e = 1.0\alpha^2 - 2.4\alpha + 1.0 \tag{10}$$

for laminar flow. Transition from laminar to turbulent flow is assumed to take place at a Reynolds number (Re) of 2300. The contraction ratios (α) are as follows for the three flow regions:

$$\alpha_l = \frac{V_\infty}{V_l}, \quad \alpha_t = \frac{V_\infty}{V_t}, \qquad \alpha_h = \frac{s_h}{s_h + t_h} \tag{11}$$

where V_∞ is the average upstream duct velocity, s_h is the fin gap width, t_h is the fin thickness and the subscripts are as defined before.

The frictional pressure drop in each region is calculated as

$$\Delta P_f = \frac{4Lf}{d_h} \times \frac{1}{2}\rho V_h^2 \tag{12}$$

from Kakaç et al. [9] where the Fanning friction factor (f) is as follows for hydrodynamically developing laminar flow

$$f = \frac{1}{Re}\left(\frac{3.44}{\left(x^+\right)^{1/2}} + \frac{24 + 0.674/\left(4x^+\right) - 3.44/\left(x^+\right)^{1/2}}{1 + 0.000029\left(x^+\right)^{-2}}\right) \tag{13}$$

and the hydrodynamic development length (x^+) is

$$x^+ = \frac{L}{Re \times d_h} \tag{14}$$

The hydraulic diameter is assumed to be twice the fin gap ($d_h = 2s_h$) as in the case of infinite parallel plates and is used as such for all calculations. Hydrodynamically developing flow was modelled because typical heat sinks operate in the developing flow region due to their typical length. As the length of the heat sink increases, the Fanning friction factor approaches the developed flow value.

The stagnation effects are treated as losses in the bypass region and are given as

$$\Delta P_s = C_d \times \frac{1}{2}\rho\left(V_\infty^{\,2} - V_h^2\right) \tag{15}$$

where C_d is 0.2 for tip or lateral bypass and 0.1 for combined bypass in the expansion region and 0.8 for contraction according to Butterbaugh and Kang [7].

The flow leakage through the fin tips causes a longitudinal velocity gradient. The fin flow velocity decreases along the length of the heat sink and the effect is more pronounced with heat sinks more densely packed with fins. Heat transfer coefficients are based on the flow velocities and the geometry of the heat sink. Due to limited space, these will not be included here.

4 Results and Discussion

A set of results obtained from Butterbaugh and Kang [7] (B&K) has been used to verify the bypass model used in this study. The physical set-up,

Transactions on Engineering Sciences vol 20, © 1998 WIT Press, www.witpress.com, ISSN 1743-3533

shown in figure 3 consists of a bonded heat sink with high aspect ratio fins as well as an adjustable flow duct.

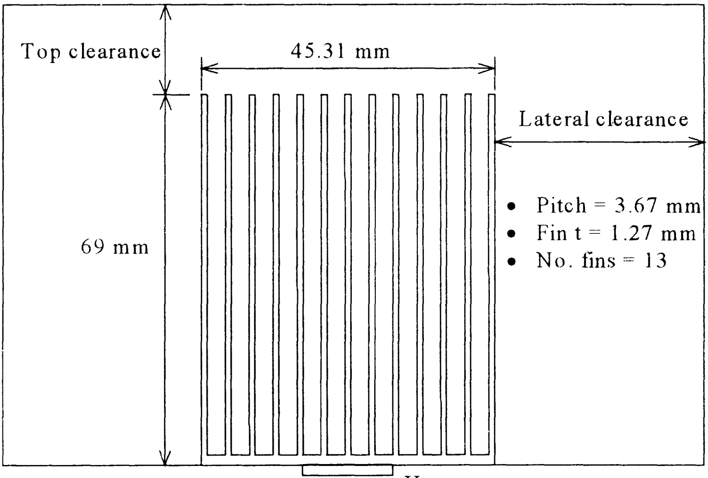

Figure 3. Flow Bypass Assembly (Butterbaugh and Kang, 1995)

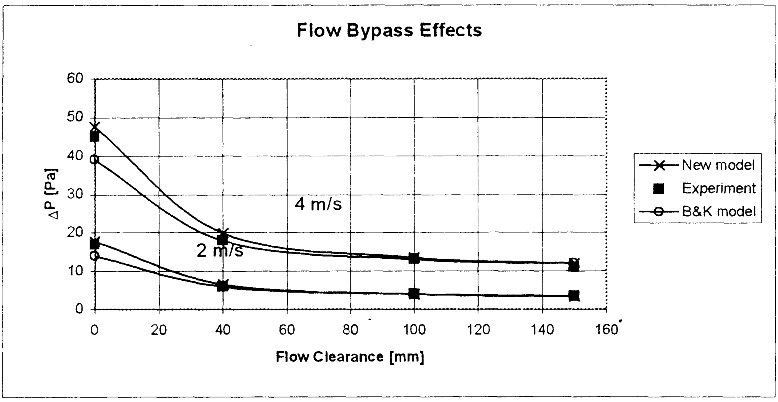

Figure 4. Comparison for Bypass Flow Pressure Drop

Figure 4 shows the results of the pressure drop across the heat sink. The pressure drop results give a good indication of the accuracy of the method, as the pressure drop balance is a primary equation in the model.

A commercial CFD program, Flotherm, was used to investigate the flow characteristics through and around a typical heat sink. The CFD analysis will be briefly discussed using one example analysis. The idea is to identify the cause and nature of tip bypass in heat sinks. At this stage, two parameters have been considered; the velocity range, which includes laminar and turbulent flow, as well as the length of the heat sink in the

Transactions on Engineering Sciences vol 20, © 1998 WIT Press, www.witpress.com, ISSN 1743-3533

flow direction. Figures 5 and 6 show the results of the CFD analysis for the longitudinal (X) and tip (Z) velocities as well as the normalized velocities in both directions. The normalized values have been based on the free stream velocity.

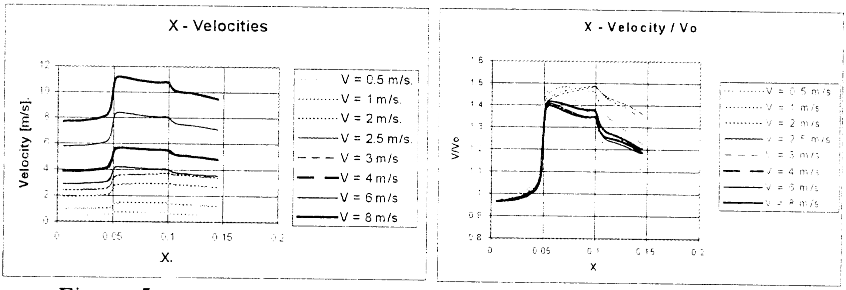

Figure 5: Longitudinal and Normalized Velocity through Heat Sink

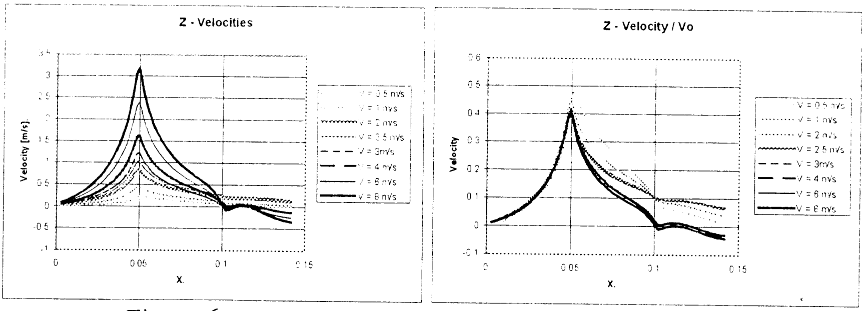

Figure 6: Tip and Normalized Tip Velocity in Heat Sink

The dimensions of the heat sink considered here are as follows: Heat sink length = 50 mm; Fin Height = 20 mm; Fin Thickness = 2.5 mm; Base Thickness = 4 mm; Fin Gap = 5 mm and the Tip Clearance = 24 mm. The computational model extends 50 mm before and after the heat sink. The velocities were measured at a symmetry point between the fin at the fin tips. The normalized values collapse well in the turbulent region but there appears to be little relation in the case of laminar flow. It is interesting to note that the longitudinal velocities at the tip of the fins decrease along the heat sink in the turbulent cases but for the laminar cases, this velocity increases. This is due to the lower velocity cases possessing less momentum and causing more air flow higher up in the heat sink. The nature of the normalized tip velocity shows this again. In the turbulent case, the sudden contraction seems to have the most influence regarding bypass whereas the laminar flow continues to have an upward velocity throughout the region.

5 Conclusions

A combination of numerical and analytical methods was combined to determine the thermal performance of extruded, forced convection heat sinks. The speed and accuracy of solutions for a limited range of setup suggests that this method can be used successfully in the design for electronics cooling. An important component of the heat transfer is the nature of the flow in and around the heat sink. An analytical model developed to determine the extent of the flow bypass in heat sink configurations was successful for bypass without tip leakage. Much work is still needed in the development of a complete model. This model would include the tip leakage and would therefore include a velocity gradient within the finned region of a heat sink.

References

1. Parry, J.D. & Rosten, H.I., A Methadology for Thermal Characterisation of IC Packages, Parts 1 and 2, *Proceedings of the Pacific Rim/ASME International Intersociety Electronic & Photonic Packaging Conference, INTERPack '97*, Hawaii, pp. 989-1006, 1997.
2. Van de Pol, D.W., and Tierney, J.K., "Free Convection Heat Transfer from Vertical Fin Arrays", IEEE Transactions on Parts, Hybrids and Packaging, Vol. PHP-10, No. 4, pp. 267-271, 1974.
3. Lee, S., "Optimum Design and Selection of Heat Sinks", *Proceedings of the 11th Annual IEEE Semi-Therm Symposium*, pp. 48-52, 1995.
4. Kim, S.J. & Lee, S., On Heat Sink Measurement and Characterization, *INTERPack '97*, Hawaii, pp. 1903-1909, 1997.
5. Belady, C., Standardizing Heat Sink Characterization for Forced Convection, *ElectronicsCooling Magazine*, pp.24-26, Sept. 1997.
6. Obinelo, I.F., Characterisation of Thermal and Hydraulic Performance of Longitudinal Fin Heat Sinks for System Level Modeling Using CFD Methods, *Proceedings of the Pacific Rim/ASME International Intersociety Electronic & Photonic Packaging Conference, INTERPack '97*, Hawaii, pp. 1239-1249, 1997.
7. Butterbaugh, M.A., and Kang, S.S., "Effect of Airflow Bypass on the Performance of Heat Sinks in Electronic Cooling", *Advances in Electronic Packaging*, EEP-Vol. 10-2, ASME 1995.
8. Kays, W.M., "Loss Coefficients for Abrupt Changes in Flow Cross Section With Low Reynolds Number Flow in Single and Multiple Tube Systems", *Transactions of the ASME*, pp. 1067-1074, November 1950.