A computer model to simulate heat transfer in heat sinks
J.A. Visser, P. Gauché

Department of Mechanical and Aeronautical Engineering, University of Pretoria, Pretoria 0002, South Africa

Abstract

The reduction in size of modern electronic components and the accompanying reduction in size of computer casings requires optimized thermal designs for reliable electronic systems. Cooling problems related to electronic systems can be divided into the cooling of components mounted on heat sinks, components mounted on boards and environmental-level analysis.

This paper will concentrate on the cooling of components with heat sinks using a finite difference simulation of the heat transfer in typical heat sink configurations. Various heat sinks were simulated to obtain the effect of the influencing parameters on the heat transfer at the boundaries. The simulated results were compared with experimental values over a range of heat dissipation in each type of heat sink. From the experiments, improved heat transfer models were developed at the boundaries to compensate for the influencing parameters such as heat sink orientation relative to gravity and by-pass air flow.

It can be concluded from the results that three-dimensional simulations have distinct advantages over traditional design graphs to analyze the performance of heat sinks and should be common practice for the electronics design engineer.

1 Introduction

The trend in modern electronics is to increase the speed and power of processing and to reduce the size and weight of electronic hardware. High density components such as multi-chip modules (MCM’s) produce significant amounts of heat resulting from increasing number of devices and power requirements. Due to these factors, the temperature at the junction of components reach critical levels which cause failure to occur. Current estimates
indicate that 55% of all electronic failure occurs due to overheating.\(^2\) One method to reduce the temperature of electronic components is to use convection heat spreaders called heat sinks which provide a large surface area for cooling.\(^3\) The fundamental objective is to keep the semi-conductor junction temperature below the critical limit, usually below 100°C to enable reliable operation.

For many years, experimental methods were used to obtain thermal data on various heat sinks. Over the years these experimental results were used by manufacturers to develop design graphs for typical heat sinks. These design graphs are, however, limited in application since they are only applicable to specific size and lengths of heat sinks.

An attractive alternative is the use of three-dimensional numerical solutions for these complex heat transfer problems. This enables the designer of a specific heat sink combination to obtain detailed information about the heat transfer in the heat sink, the heat transfer coefficients and thermal resistances. At the same time, new designs can be evaluated in a cost effective way. More complex heat transfer coefficients can also be introduced to improve the accuracy of simulations.

A literature survey has shown an increasing interest and awareness in the field of electronics cooling. Kays\(^4,5\) extensively researched compact heat exchangers at an early stage. His analytical and experimental results are commonly used in current work. Knight et al.\(^3,6,7\) developed generalized, non-dimensional optimization techniques for sizing coolant channels for heat sinks. Azar and co-workers\(^8\) investigated narrow channel heat sinks with top clearance versus ducted systems to determine the effects on the heat transfer. Bypass has subsequently been investigated by Lee\(^1\) and Butterbaugh and Kang\(^9\) making use of analytical techniques to determine fin flow velocity. From a numerical perspective, George et al.\(^10,11\) has developed methodologies in automatic mesh generation making numerical heat transfer research more viable. Numerical heat transfer is often based on methods reported by Patankar\(^12\).

The thermal problem considered is often referred to as a thermal resistance circuit and is analogous to it’s electrical resistance counterpart\(^1\). Figure 1 shows the nodes in a typical resistance circuit found in electronic assemblies. The overall thermal performance can be measured by the total assembly thermal resistance \((R_{\text{assembly}})\) which comprises of: The junction-to-case resistance \((R_{jc})\), The case-to-sink resistance \((R_{cs})\); and the sink-to-ambient resistance \((R_{sa})\).

\[
R_{\text{assembly}} = R_{jc} + R_{cs} + R_{sa}
\]  

(1)

The thermal resistance is defined by the temperatures between two points \((\Delta T)\) and the rate of heat dissipation \((Q)\) so that:

\[
R = \frac{\Delta T}{Q}
\]

(2)
This paper outlines the method of predicting three-dimensional heat transfer and resulting thermal parameters in electronic assemblies containing heat sinks. Predicted and measured data is also discussed.

2 Modes of Heat Transfer

The thermal resistance circuit encountered in an electronic assembly is influenced by five forms of heat transfer, namely: Heat conduction from the junction to case of the electronic component; Heat conduction from the case to heat sink through the bonding layer; Heat conduction through the heat sink; Convection to the surrounding cooling medium; and Radiation to the surroundings. These heat transfer forms are shown in figure 1.

3 Outline of Modeling Procedure

3.1 Governing and finite difference equations

The partial differential equation that describes the heat transfer in an extruded aluminium heat sink is the three-dimensional unsteady conductive heat transfer equation also known as Poisson's equation. In Cartesian co-ordinates this equation is

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

(3)
Advanced Computational Methods in Heat Transfer

The finite difference equations used in the numerical procedure are derived by integrating the partial differential equation over control volumes surrounding a grid point. The general finite difference equation for eqn (3) can be written as

$$a_p T_p = \sum a_{np} T_{np} + b$$

where the coefficients $a_p$ at the central grid point and $a_{np}$ at the neighboring points depend on the material properties and grid dimensions used in the solution procedure. At the boundaries of the extruded heat sink the term $b$ and the coefficient $a_p$ in eqn (4) contain information on the convective and radiative heat transfer from the heat sink.

3.2 Boundary conditions

During operation, heat is dissipated from the heat sink and component through convective and radiative heat transfer from the heat sink and component boundaries. These heat transfer processes are accounted for in the modeling procedure by means of certain boundary conditions described in this section.

The general finite difference equation (4) is also valid at the convective and radiative boundaries of the heat sink. The coefficient $a_p$ contains information on the convective and radiative heat transfer coefficients, while the term $b$ contains information on the surrounding fluid temperature and the heat transfer coefficients. The heat transfer at the boundaries can therefore be accounted for by using known fluid and surrounding temperatures and by calculating heat transfer coefficients for convection and radiation.

3.2.1 Forced convection boundaries

Forced convection is the term used to describe the heat transfer from a surface to a moving fluid where the fluid is pumped or propelled over the cooling surface from another source.

The forced convection heat transfer coefficient between the heat sink and the surrounding air is calculated by assuming that the heat sink can be approximated as flat plates. For a flat plate the mean coefficient $h_{ca}$ can be calculated from the equation

$$h_{ca} = \text{constant} \times Pr^{1/3} Re^{1/2}$$

This is given as

$$h_{ca} = 3.785 \sqrt{\frac{V}{L}}$$

by Ellison for heat sinks. $Pr$ is the Prandtl number and $Re$ is the Reynolds number. $V$ is the fin flow velocity in [m/s] and $L$ is the length of the finned region. Ellison also proposes a modification factor, $C$ to account for turbulence caused by fins. This is given as:
\[ C = 0.33(\log V) + 1.467 \quad \text{for} \quad V < 5.1 \text{m/s} \]  \hspace{0.5cm} (7)
\[ C = 1.00(\log V) + 0.994 \quad \text{for} \quad V \geq 5.1 \text{ m/s} \]  \hspace{0.5cm} (8)

The fin flow velocity, \( V \) is calculated if necessary by using a flow bypass model. This is used when approach velocity, \( U \) is known and the flow is forced through a semi-restricted or fully-restricted system, i.e. with and without bypass respectively, typical of most electronic systems. An analytical model proposed by Butterbaugh and Kang\(^9\) was adapted for the purpose of this study. For a given approach velocity and known duct dimensions, the air flow rate, \( Q \) is used to determine the flow velocity in the finned region by applying pressure balance and mass conservation conditions. This takes into account the area ratio of the heat sink, contraction and expansion losses in all regions, stagnation effects and frictional pressure drops.

### 3.2.2 Natural convection boundaries

Natural convection is driven by buoyancy forces arising from density differences due to the heated fluid, as in the case of natural-draft cooling towers. Van de Pol and Tierney\(^15\) developed an empirical correlation for vertical U-channel geometries and is given as

\[ Nu_r = \frac{Ra^*}{\psi} \left[ 1 - \exp\left[ -\psi\left(\frac{0.5}{Ra^*}\right)^{\frac{3}{4}}\right] \right] \]  \hspace{0.5cm} (9)

where \( Nu_r \) is the Nusselt number, \( Ra^* \) is the Raleigh number,

\[ \psi = \frac{24\left(1 - 0.483e^{-0.17a}\right)}{\left[1 + \frac{a}{2}\right]\left[1 + \left(1 - e^{0.83a}\right)(9.14a^{1/2}e^{45} - 0.61)\right]^3} \]  \hspace{0.5cm} (10)

and

\[ Ra^* = \frac{(r/H)GrPr}{r} \]  \hspace{0.5cm} (11)
\[ r = \frac{2LS}{(2L + S)} \]  \hspace{0.5cm} (12)
\[ a = \frac{S}{L} \]  \hspace{0.5cm} (13)
\[ V = -0.30 \quad [m^{-1}] \]  \hspace{0.5cm} (14)
\[ Gr = \frac{g\rho^2}{\mu^2} \beta(T_b - T_o)r^3 \]  \hspace{0.5cm} (15)
574 Advanced Computational Methods in Heat Transfer

\[ Pr = \frac{\mu C_p}{k} \]  

(16)

\( r \) is the hydraulic radius. \( S, H, L \) and \( a \) are heat sink parameters. \( V \) is spatial variable and \( Gr_r \) is the Grashof number.

The average heat transfer coefficient can be calculated as

\[ \overline{h_{ca}} = \frac{k}{r} \frac{Nu_r}{r} \]  

(17)

3.2.3 Radiation boundaries

With a radiative heat transfer coefficient, the effect of radiation heat losses from the boundary of the heat sink can be treated in the same way as the convective heat transfer. A heat sink is treated as a solid body with dimensions the same as the overall heat sink dimensions and the radiative heat transfer coefficient is defined as

\[ h_r = \varepsilon_r \sigma \left( T_h^2 + T_s^2 \right) \left( T_h + T_s \right) \]  

(18)

where \( T_h \) is the surface temperature of the heat sink and \( T_s \) is the temperature of the surrounding environment and is assumed to be equal to the air temperature.

The radiative heat transfer coefficient is then modified to account for radiation from within the finned region and also by a ratio for the heat sink surface area versus the solid body area to obtain \( \overline{h_r} \).

3.3 Solution of the heat transfer in heat sinks

3.3.1 Grid generation

A structured grid generation scheme using algebraic interpolation techniques is used for this study. Boundary conforming curvilinear grids are generated by separating the heat sink into rectangular blocks in the computational domain and then combining these to form a multiply connected grid. This technique is usually referred to as the multi-block method.

As with any numerical simulation, the solution grid needs to have a sufficient number of grid cells to reach an accurate answer. By progressively increasing the number of grid cells for a given case, a grid independent solution can be achieved which should then give the desired result. It was found that the number of grid cells needed in this study could be relatively small to give satisfactory results as the high conductivity of heat sinks provides a solution with low thermal gradients.

3.3.2 Solution procedure

The governing equation is transformed to be used in a general curvilinear coordinate system and then discretised to form a linear system of equations that are solved to obtain a final solution. The Gauss-Seidel iteration scheme is used with Successive Over-Relaxation (SOR). This algorithm iteratively steps
Advanced Computational Methods in Heat Transfer

through the solution field updating the temperature at each element based on the values of its neighbors. The convergence rate or residual is used as a criterion to determine if the solution is sufficiently converged.

3.3.3 Stability, convergence and computing effort
The governing equation is unconditionally stable and no problems are expected when SOR values within limits are used.

A typical grid-independent solution of a heat sink and component containing 1056 grid cells was attained in 46 seconds using an IBM compatible 486 DX2 machine and the C++ programming language. This converts to a rate of 2.44E-4 seconds per iteration per grid cell.

4 Outline of Experimental Procedure

The experimental work was done using a purpose built testing frame which is capable of mounting a heat sink in any of the possible orientations encountered in electronic enclosures. A PC73A™ card with cold junction compensation was used to log the measured temperatures into a voltage file. The card has an accuracy of ±1.0 °C. The ambient was measured with a sensor mounted on the auxiliary card. The card was calibrated using standard procedures. Four type J, 1.5 mm diameter, grounded thermocouples manufactured to NBS(NIST)/ANSI standards were used for the temperature measurements and were connected to the PC73A card. Four points on a selected heat sink were chosen for measurements to get an indication of the accuracy of the solution at discrete points on the heat sink. A Topward 6063D™ 30V/6A dual digital power supply was used to supply a load to the components. Accuracy is 0.1 Volts and 0.01 Amperes.

A chosen heat sink and component assembly was tested by recording the temperatures at four points for sufficient time to reach steady state conditions. This was done for a number of orientations relative to gravity for each assembly tested. The assembly constructed for this study consisted of an aluminium heat sink with 20 fins. Each fin was 4 mm thick, 30 mm tall and 96 mm long in the flow direction. The base was 10 mm thick and 200 mm wide with the fins evenly spaced. A TO-3 casing with transistor was mounted at a discrete location on the base using silicon heat transfer compound. The contact thermal resistance was approximated as 0.05 °C/W.

5 Results and Discussion

Selected results are given and discussed. For natural convection, results are based on the assembly described in the last section. Figure 2 compares the experimentally measured temperature versus the predicted temperature at a point directly next to the component with the ambient as the reference temperature. For all the applied power ratings, the data compares very well with
the predicted performance with a maximum difference of 6.19 % between experimental and predicted results. Similar results were obtained for various other orientations with respect to gravity. Longer lengths of heat sink also gave good results except for the case of a 286 mm heat sink which gave a maximum difference of 13.13 % in one orientation and 30.6% in another orientation.

In the case of forced convection, a set of data obtained from Gopalakrishna was used. The author used a specialized heat sink with two components and tested the assembly at 0.5, 1.0, 2.0 and 4.0 m/s experimentally. Figure 3 shows a comparison of measured and predicted results. The maximum difference between the forced convection model and experimental values was 12.0 % for the 1.0 m/s case. For the other flow velocities the maximum difference was less than 5.0 %.

---

**Theoretical vs. Experimental Results**

**Natural Convection, Heat Sink Fins and Base-Vertical**

![Graph showing theoretical vs. experimental results for natural convection](image)

**Experimental vs. Theoretical Results**

**Forced Convection, Various Fin Flow Velocities**

![Graph showing experimental vs. theoretical results for forced convection](image)
For the forced convection case the thermal resistance for a device power of 3 W is shown in figure 4. This shows the typical tendency of the thermal resistance to decrease with increasing fin flow velocity.

![Thermal Resistance for Forced Convection](image)

Figure 4: Thermal resistance for the forced convection case

### 6 Conclusions

A numerical model to solve the heat transfer in electronic components connected to heat sinks has been developed. A heat sink and component assembly can be generated using an automatic grid generation process providing a suitable grid for the solving process. Output is in the form of detailed temperature information, maximum temperatures, thermal resistances and heat sink efficiency.

Experimentation has been conducted to improve and verify the heat transfer model. Results show very good agreement for most cases, especially for heat sinks of simple geometries.

Further work involves more experimentation, especially for forced convection cases with both bypass and fully restricted flow paths. Improved heat transfer models are needed, particularly for irregular heat sink geometries, for example where fin gap and fin height are not consistent.

The results produced show that this model has distinct advantages over traditional design methods for heat sinks and should become common practice for the electronic design engineer.

### References


