A computational method to design heat sinks

J.A. Visser

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

Abstract

A computationally economical method for the design of aluminum heat sinks is presented. The motivation for this project arose from a growing need in the electronics industry to find efficient ways of improving the design of heat sinks. A three-dimensional finite-difference simulation using curvilinear boundary fitted grids is used to model the conduction heat transfer in the heat sink. Analytical and empirical models for the boundary conditions take into account the effects of natural or forced convection, as well as radiation to the surroundings. Flow bypass is included in the analysis for forced convection. The solution methodology enables solution times of less than 30 seconds for most heat sink geometries. The quick analysis and accurate results verified by experimentation indicate that the techniques developed and implemented can be used in the design and selection of heat sinks.

Nomenclature

\( \text{ar} \) Heat sink channel aspect ratio \( A \) Area \( [m^2] \)
\( c \) Specific heat \( [J/kgK] \) \( d_h \) Hydraulic diameter \( [m] \)
\( Gr \) Grashof number \( H \) Height of heat sink \( [m] \)
\( h \) Heat transfer coefficient \( [W/m^2K] \) \( k \) Conductivity \( [W/mK] \)
\( L \) Length of heat sink \( [m] \) \( m \) Mass flow rate \( [kg/s] \)
\( \bar{Nu} \) Mean Nusselt number \( P \) Pressure \( [Pa] \)
\( Pr \) Prandtl number \( Q \) Volume flow rate \( [m^3/s] \)
\( Ra^* \) Modified Raleigh number \( Re \) Reynolds number
\( R \) Thermal resistance \( [K/W] \) \( S \) Source term \( [W/m^3] \)
\( T \) Temperature \( [K] \) \( V \) Velocity \( [m/s] \)
564 Advanced Computational Methods in Heat Transfer VI

<table>
<thead>
<tr>
<th>W</th>
<th>Width of heat sink [m]</th>
<th>( \rho )</th>
<th>Density [kg/m³]</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \beta )</td>
<td>Expansion Coefficient [K⁻¹]</td>
<td>( \epsilon )</td>
<td>Emissivity of heat sink</td>
</tr>
</tbody>
</table>

1 Introduction

The trend in modern electronics is to increase the speed and power of processing and to decrease the size and weight of electronic hardware. This increasing power demand can cause the temperature at the die junction of a component to reach a critical level, above which failure may occur. Current estimates indicate that 55% of all electronic failures occur due to overheating caused by these high power devices [1]. One method to reduce the temperature of electronic components is to use convection heat spreaders, commonly called heat sinks, which provide a large surface area for cooling.

For many years, experimental methods have been used to obtain thermal data for the heat sinks. These experimental results have then been used by manufacturers to develop design graphs for heat sink applications. Design graphs are however limited in application, since they are only applicable to specific size, length of the heat sink tested and specific boundary conditions arising from channel configuration and flow conditions. An attractive alternative is the use of three-dimensional numerical methods to solve these complex heat transfer problems. This enables the designer of a specific heat sink, or combination of heat sink and electronic assembly, to obtain detailed information about the relevant heat transfer, heat transfer coefficients and thermal resistances. The use of CFD techniques to design new heat sinks, however, remains a time consuming task that often frustrates the design engineer.

The requirement of a design engineer is a heat sink design program that can predict the thermal performance of heat sinks in seconds, while still being able to change the geometry of the heat sink and the environmental conditions quickly. This paper outlines such a design program. The method differs from CFD in that the fundamental mass, momentum and energy equations are not solved for in the flow regions. Only the conduction in a heat sink is solved using the discretised curvilinear diffusion equation with a numerical grid to represent the heat sink. This provides an accurate solution for the thermal gradients and consequently the overall heat sink thermal resistance and efficiency. The effect of the environment on the heat sink is modelled by using a combination of a network analysis and empirical boundary conditions.

In recent years a fair amount of research have been conducted on the different sub-sections of this concept. Knight et. al. [2] developed generalised, non-dimensional optimisation techniques for sizing coolant channels for heat sinks. Azar et. al. [3] 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 [4] as well as Butterbaugh and Kang [5] making use of analytical techniques to determine fin flow velocity. Bekker [6] described the discretisation of the conduction equation in curvilinear co-ordinates and introduced an acceleration method to speed up convergence.
2 Governing equations

The thermal problem associated with the analysis of heat sinks is often referred to as a thermal resistance circuit and is analogous to its electrical resistance counterpart. Figure 1 shows a typical resistance circuit found in heat sink assemblies. The thermal resistance is defined by the temperatures between a point on the heat sinks and the temperature on the environment (ΔT) and the rate of heat dissipation (Q_H) so that:

\[ R = \frac{ΔT}{Q_H} \]  

Figure 1: Thermal resistance circuit and heat transfer mechanisms.

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

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

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_{max}) for the definition of the thermal resistance (R_{th}). 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 heat transfer to the environment accounts for both convection and radiation as shown in equation 3.

\[ Q_H = (h_{conv} + h_r) A (T_h - T_r) \]
The boundary conditions are specified using compact analytical and empirical models. Van de Pol and Tierney [7] developed an empirical correlation for vertical U-channel geometries exposed to natural convection and is given as:

\[
\mathrm{Nu}_r = \frac{Ra^*}{\psi} \left[ 1 - \exp \left( -\frac{0.5}{\psi \left( Ra^* \right)} \right) \right]^3
\]  

(4)

where

\[
\psi = \frac{24 \left( 1 - 0.483e^{-0.175
\rho
k}
\right)}{(1 + \epsilon/2) \left[ 1 + \left( 1 - e^{-0.83\epsilon/\rho} \right) \left( 0.14\epsilon^{1/2} e^{\epsilon/\rho - 0.61} \right) \right]^3}
\]  

(5)

The forced convection heat transfer coefficient between the heat sink and the surrounding air is calculated by using the analytical formulation:

\[
\overline{h}_{\text{conv}} = CF \times 3.455 \sqrt{\frac{\rho V_h}{L}}
\]  

(6)

where \( CF \) is a modification factor used to account for turbulence caused by the fins.

With a radiation 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 cuboid body with dimensions the same as the overall heat sink dimensions and the radiation component is derived from the general radiation model:

\[
h_r = \varepsilon \sigma (T_h + T_r) \left( T_h^2 + T_r^2 \right)
\]  

(7)

where \( T_h \) is the surface temperature of each element in the heat sink and \( T_r \) is the temperature to which the heat sink radiates.

An analytical model proposed by Butterbaugh and Kang [5] was adapted and used to determine the flow velocities around a heat sink. 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. Each flow path has to conform to a pressure drop (\( \Delta P \)) balance and mass conservation laws so that

\[
\Delta P_h = \Delta P_i = \Delta P_f \quad \text{and} \quad Q_h + Q_i + Q_f = Q
\]  

(8)

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 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.
3 Verification and discussion

An important component in the development of a methodology for predicting the temperature distribution in heat sink assemblies, especially when empirical formulations are used to represent non-linear field properties, is a broad scope of experimentation or other verification. In this paper verification was based on results obtained with the Qfin 2.0 program and compared to experimental data and in some cases results obtained from CFD simulations using the Icepack program. Verification includes natural convection and forced convection cases for conventional and pin fin heat sinks exposed to the effect of flow bypass in a channel.

3.1 Natural convection

Heat sinks typically used in natural convection environments have short, tapered fins, relatively wide fin spacing and a wide, thick base. These heat sinks are usually mounted externally on the back vertical chassis of power electronic products. The heat sink profile shown in Figure 2 is typical of this description and was tested in three orientations and with various devices.

**Heat sink dimensions:**
- 9 Aluminum fins: 1.0 mm thick, 8.8 mm high, spaced 5.8 mm apart.
- 3 Aluminum fins: 1.0 mm thick, 5.5 mm high, spaced 3.7 mm apart on each side of the outer fins.
- Aluminum base: 2.0 mm thick, 76 mm wide, 50 mm long extrusion, stepped up 3.3 mm at 14 mm from the left and right edges.
- Colour: Black

**Test environment:**
- 25 mm square, 65 W heat source centred on the base.
- Natural convection
- 25°C ambient air temperature

![Figure 2: Heat sink profile used for natural convection case.](image-url)
Table 1 shows a comparison of the experimentally measured temperatures versus the predicted temperatures at a point directly underneath the component. The ambient is used as the reference temperature. For all the applied power ratings, the data compares very well with the predicted performance with a maximum difference of 2% between experimental and predicted results.

Finally, a grid dependence test was undertaken for the natural convection study. The results for a coarse grid differed very little from the results of a fine grid. In fact, the difference is hardly distinguishable. This is attributed to the small thermal gradients experienced due to the high thermal conductivity, and the boundary conditions that are identical for any grid for a given assembly.

### 3.2 Forced convection

The first forced convection case is shown in Figure 3. The configuration consists of a small heat sink placed in a fairly large duct that results in a large bypass ratio and a small pressure drop.

**Heat Sink Dimensions:**
- 5 evenly-spaced Aluminum fins 1.65mm thick, 13mm high and 76.2mm long
- Aluminum base 3.3mm thick, 25mm wide and 76.2mm long
- Colour: Shiny

**Test Environment:**
- 200mm by 200mm square duct
- Uniformly distributed heat source at the bottom of the heat sink
- 25°C inlet air temperature
Figure 3: Heat sink profile used for a large-bypass forced convection case.

Thermal resistance for natural convection:

- Experimental: \(9.6 \, ^\circ C/W\)
- CFD: \(10.8 \, ^\circ C/W\)
- QFin: \(11.3 \, ^\circ C/W\)

![Figure 4: Thermal resistance at different approach air velocities.](image)

The results shown in Figure 4 indicate good correlation with both experimental data and CFD results with both Qfin and CFD results somewhat below the experimental data toward the higher velocities. The difference is, however, small. In lower velocity area where transition is very important, both the Qfin and CFD results under predict the rate of heat transfer. The second forced convection case consist of a folded fin heat sink in a non-conformal wind tunnel as shown in Figure 5.
Heat sink dimensions:
- 40 evenly spaced Aluminum fins: 0.4 mm thick, 32.6 mm high and 125 mm long.
- Aluminum base: 5 mm thick, 100 mm wide, 125 mm long
- 0.4 mm thick shroud on the top of the heat sink
- Colour: Shiny

Test environments:
- Duct: 120 mm wide by 42 mm high
- 100W source of 100 mm square attached to the centre of the base.
- Forced convection, 1m/s to 5 m/s approach velocity.
- 40°C ambient air temperature

Figure 5: Heat sink profile for a small-bypass forced convection case.

Figure 6: Thermal resistance at different approach air velocities.

The thermal resistance predicted with Qfin is compared to experimental data in Figure 6. The results indicate good correlation with experimental data in the velocity range indicated. This confirms that the by-pass model used are able to predict the split in flow around the heat sink with an acceptable degree of accuracy. Since the same basic formulation is used to predict the heat transfer
and by-pass in pin fin heat sinks, the third forced convection case consist of a pin fin heat sink located in a square duct. The heat sink is shown in Figure 7.

**Heat Sink Dimensions:**
- 11x12 evenly-spaced rectangular Aluminum pin fins, 2.3mm x 2.24mm cross section and 21.6mm high distributed over a 53.3mm x 53.3mm base with thickness of 3.8mm.
- Colour : Shiny

**Test Environment:**
- 200mm by 200mm square duct
- Uniformly distributed heat source at the bottom of the heat sink
- 25°C inlet air temperature

![Figure 7: Heat sink profile used for pin fin test case.](image)

The thermal resistance predicted with Qfin is compared to experimental and CFD data in Figure 8. The results indicate good correlation between Qfin and CFD results, with a small deviation from experimental results. Once again the deviation from experimental data mainly occurred towards the lower flow.

![Figure 8: Thermal resistance at different approach air velocities.](image)
velocities, with both the CFD and Qfin seemingly unable to pick up the sharp increase in thermal resistance at the low velocities.

4 Conclusions

A computational method to evaluate the thermal performance of heat sink assemblies was presented. This method provides a quick and easy tool that enables the user to design and evaluate assemblies in an almost real-time process. The combination of numerical, analytical and empirical formulations makes this method many times faster than full CFD codes but still provides accurate results. This was proven by the comprehensive verification results included in this paper. 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.

5 References


