THERMAL DESIGN OF A PCB

By

STANLEY L BENTLEY, P.E.
Thermal Design of a PCB

• The role of the printed circuit board in an electronic device is continuing to evolve. The PCB is now performing the following functions:

  • The Support Structure for the electronic components
  • The interconnect means for the electronic components
  • The pathway for signals requiring the traces to emulate the AC characteristics of a coaxial cable
  • The Thermal gateway for removing heat from the electronic components.
THERMAL CONDUCTANCE

Thermal conductivity is:
"The quantity of heat transmitted through a unit thickness in a direction normal to a surface of unit area, due to a unit temperature gradient under steady state conditions"

"Conductance Is the measure of how easy it is for Stuff to flow and is the reciprocal of Resistance."

Thermal Conductance of common materials (W/(M * °C))

<table>
<thead>
<tr>
<th>Material</th>
<th>Conductance (W/(M * °C))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air</td>
<td>0.024</td>
</tr>
<tr>
<td>Glass/Epoxy</td>
<td>0.12</td>
</tr>
<tr>
<td>Glass</td>
<td>1.05</td>
</tr>
<tr>
<td>Solder:</td>
<td></td>
</tr>
<tr>
<td>(Sn/Pb, SAC305)</td>
<td>50.00</td>
</tr>
<tr>
<td>Tin (Sn)</td>
<td>67.00</td>
</tr>
<tr>
<td>Copper (Cu)</td>
<td>401.00</td>
</tr>
<tr>
<td>Silver (Ag)</td>
<td>429.00</td>
</tr>
</tbody>
</table>
Thermal Design of a PCB

• The thermal gateway is the property to be discussed. The epoxy/glass structure of a conventional PCB is a poor medium for the removal of heat. The resin has a very low thermal conductance (measured in W/M °K) resulting in a very high thermal resistance. If only the reinforced resin structure is used as a primary heat conduit, then the likely result is damage to the electronic components, damage to the PCB, or both. For this reason, other means have been developed to more efficiently conduct the heat away from the components. These means include:
  • Thinner resin structures
  • Metal backing plates
  • Layers of high thermal conductance material
  • Thermal vias
  • Rigid Flex structures
Thermal Design of a PCB

THINNER RESIN STRUCTURES

• Thin dielectrics are usually single side PCB’s
• The structural strength is reduced
• The circuitry density is poor
• The applications are usually limited to “LED Type” circuits
Thermal Design of a PCB

METAL BACKING PLATES

• Thin resin structures are often bonded to a metal backing plate
• The metal structure can allow the dielectric to be as thin as 2 mils (0.002”)
• The adhesive will be at least 2 mils in thickness and will have a thermal conductance approximately equal to the resin
• The thermal Resistance will now be the sum of the three thermal resistances
  – Primary Resin Structure + Adhesive + Metal backing plate
Thermal Design of a PCB

LAYERS OF HIGH THERMAL CONDUCTANCE MATERIAL

• To improve the thermal conductance of the “stack up” and also to increase the circuit density, multilayer structures can be created using inner-layer cores that have a higher thermal conductance than the glass/epoxy.
  – Typical values range from 0.7 to 7 (W/M °K)
• These cores can also be used with a metal backing plate, which will result in a much lower Thermal Resistance of the “stack up” than a thin Glass/Epoxy structure
• The draw back is the cost of these cores
Thermal Design of a PCB

THERMAL VIAS

• Thermal vias are a way to use conventional resin structures (glass/epoxy) and still conduct large amounts of heat away from the components.
• Thermal vias do not need to be electrically tied to the components. If they are not electrically tied, then they can be routed directly to the “heat sink” means
• If the vias are electrically tied to the components, then some additional Thermal Resistance is added due to the dielectric layer that is required.
• Thermal vias add little cost, but they do consume “real estate”.
• No combination of technologies (at this time) yields a higher Thermal conductance for the design.
Thermal Design of a PCB

RIGID FLEX STRUCTURES

- An emerging technology combines several of the other thermal means. This technology is the Rigid/Flex
- In this technology, a flex layer is inserted in a multilayer design
- The flex layer is extended beyond the perimeter of the rigid board
- The extended section is thermally attached to a heat sinking means.
- The technology is very stable, having been used many years to create an interconnecting means for rigid boards
- The cost is dropping, but they are still much more expensive than conventional multilayer boards.
Thermal Design of a PCB

STANDARDIZED THERMAL DESIGN MODEL

The variables included in a calculation of the thermal efficiency are:
• Thermal conductance of each material in the heat path (w/m-\(0^o\)K)
• The thickness of each material in the heat path (m)
• The surface area of the heat path (m\(^2\))
• The Thermal Resistance of the heat path (\(0^o\)K/Watt)
• The heat generated across the surface area (Watts)
• The desired temperatures for top side (components), thermal path (the PCB and support structure), and the heat sinking surface (usually the maximum ambient temperature)
• The maximum allowable cost for the thermal structure
STANDARDIZED THERMAL DESIGN MODEL

To simplify this process, the author has created a PCB Standard that can be used to quickly determine a design solution. We know that all materials have a Thermal Conductivity. The insulating materials used in PCB fabrication vary from 0.1 to 7.5 (W/M°K).

First, thermal conductance must be converted into a Thermal Resistance of the Insulating Material (TIM). To do this several variables must be converted into constants. For this purpose, a circle of Radius 1 was chosen.

This gives an area of 3.14 in².

The circular pattern has the property of removing placement geometry from the calculation and providing a uniform distribution of the heat.

As the design evolves into an actual product profile (outline), the author recommends that the calculations still use increments of the 1 inch radius. This will show potential “hot spots” in the design.
STANDARDIZED THERMAL DESIGN MODEL

• The LED PCB Standard Thickness table is reduced to one constant and four variables.
  • The Constant is:
  • 3.14 Square inches of surface area

• The variables are:
  • Thermal Conductance of the System
    – This is a figure that can be obtained from a data sheet for the selected Dielectric Insulation Material
  • Total Watts of power created by the circuit elements
    – For larger surfaces, the “Total Watts” must be divided by the ratio of the area to 3.14 (in in²)
  • Thickness of the Thermal Structure.
    – Materials with high Thermal Conductance are very thin and must be reinforced with a backing material (typically Aluminum or Copper)

• Temperature drop that can be allowed across the Thermal Structure
  – This is called,” TIM”, which stands for Thermal Insulating Material.
  – From the temperature drop, the designer can determine the maximum component temperature by adding the maximum ambient temperature to the temperature drop across the TIM.
Circle of radius 1 using varying thickness of Glass/Epoxy

<table>
<thead>
<tr>
<th>Thcond</th>
<th>TIM_{th} Inch</th>
<th>R_{th} (°K/W)</th>
<th>ΔT (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.12</td>
<td>0.656167</td>
<td>25</td>
<td>50</td>
</tr>
<tr>
<td>0.0020</td>
<td>0.98425</td>
<td>25.40</td>
<td>50.80</td>
</tr>
<tr>
<td>0.0030</td>
<td>1.312333</td>
<td>19.05</td>
<td>38.10</td>
</tr>
<tr>
<td>0.0040</td>
<td>3.280833</td>
<td>7.62</td>
<td>15.24</td>
</tr>
</tbody>
</table>

WATTS
CONDUCTANCE APPLIED TO A PCB

Conductance

• Is the measure of how easy it is for heat to flow and is the reciprocal of thermal Resistance.
• We know the Thermal Conductance of a glass epoxy PCB is almost entirely through the copper vias
• In a double side PCB, there is only one flow path
• In a multilayer board, there are multiple flow paths.
  – A primary path through the via holes and
  – Additional paths at each layer that has a connection to the primary path.
**THERMAL CONDUCTANCE**

**Thermal conductivity** is:
”The quantity of heat transmitted through a unit thickness in a direction normal to a surface of unit area, due to a unit temperature gradient under steady state conditions”

Conductance is the measure of how easy it is for Stuff to flow and is the reciprocal of Resistance.

**Thermal Conductance of common materials (W/(m * °C))**

<table>
<thead>
<tr>
<th>Material</th>
<th>Conductance (W/(m * °C))</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air</td>
<td>0.024</td>
</tr>
<tr>
<td>Glass/Epoxy</td>
<td>0.12</td>
</tr>
<tr>
<td>Glass</td>
<td>1.05</td>
</tr>
<tr>
<td>Solder:</td>
<td></td>
</tr>
<tr>
<td>(Sn/Pb, SAC305)</td>
<td>50.00</td>
</tr>
<tr>
<td>Tin (Sn)</td>
<td>67.00</td>
</tr>
<tr>
<td>Copper (Cu)</td>
<td>401.00</td>
</tr>
<tr>
<td>Silver (Ag)</td>
<td>429.00</td>
</tr>
</tbody>
</table>
COPPER “WEIGHT” (Thickness)

The finished copper thickness on the surface of a PCB is the sum of (at least) two processes.

• The fabricator starts with a beginning foil weight, whose thickness is expressed in “ounces”
• The fabricator electro plates one or more additional layers of copper whose thickness is expressed in “mils” (.001”)

• The end result is a surface copper thickness that is confusing to determine.

The “ounces” designation comes from one ounce (by weight) of copper uniformly spread over 1 square foot of surface.

• The resulting thickness is 1.4 Mils (0.0014”)
• Therefore, when one hears, “one ounce” the meaning is that the thickness of the copper is 1.4 mils.
• The progression is linear. Two ounces would be 2.8 mils thick and so on.
COPPER “WEIGHT” (Thickness)

Under cross section analysis, the layers of copper are very evident. The boundary points are called, “Knit Lines”. A typical plated through PCB has a surface copper that is the sum of (1.4 mils of base copper + 100 micro inches of electroless copper + 1 mil of electroplated copper = 2.5 mils of thickness)

• The thickness of this copper is very critical to the heat removal process.
  • Too thin and it could be dissolved during lead-free processing. This is called “Copper Dissolution”. No-Lead solders are much more aggressive in dissolving copper.

• Additionally, the plated copper barrel, must be able to withstand the forces that are created by thermal expansion. There is a large mismatch between the Coefficient of Thermal Expansion (CTE) of the laminate and the copper plated barrel of the thru-hole.
Water Tower Analogy of Heat Flow in DSB

Height of tower is analogous to the temperature of the components

The Pipe is analogous to the PTH Barrel.

Reduce dia Pipe size will limit water flow

Water flow is analogous to Heat flow
Multi-Layer Analogy of divided Heat Flow

Add more buckets and Reduce the Flow
CALCULATING THERMAL CONDUCTANCE

Thermal conductivity is the quantity of heat transmitted through a unit thickness in a direction normal to a surface of unit area, due to a unit temperature gradient under steady state conditions

- $1 \text{ W/(m.K)} = 1 \text{ W/(m.}^\circ\text{C)} = 0.85984 \text{ kcal/(hr.m.}^\circ\text{C)} = 0.5779 \text{ Btu/(ft.hr.}^\circ\text{F)}$

- $q/A = k \Delta T/s$ (Steady state version of Fourier’s law)
- $q/A = \text{heat transfer per unit area (W.m}^2\text{)}$
- $k = \text{thermal conductivity (W/m K)}$
- $\Delta T = \text{temperature difference (}^\circ\text{C)}$
- $s = \text{PCB thickness}$

Copper Wall
- 0.001” wall thickness (25.4 * 10^{-6} M)
- .062” PCB Thickness (1.57 *10^{-3} M)
- 100 °C Temperature Difference
- (400 W/m. °C * .5779 = 231.16 BTU/ft . Hr . °F)

- $q/A = 25,470 \text{ KW/m}^2$

- 1 "mil" of copper in an .040" hole has a cross sectional area of $4 \times 10^{-8} \text{ M}^2$

Therefore the conductance (heat flow) through this area is:

$25,470 \text{ KW/M}^2 \times 4 \times 10^{-8} \text{ Which is equal to 1.09 Watts}$
In a Double Side PCB, there is only one flow path for heat (through the hole).

The copper wall thickness is very important for the heat to flow.

1 mil of copper in 40 mil via will conduct 1 Watt.

The Cross Sectional Area of Copper barrel determines the Heat Flow.

A copper barrel 0.040” in diameter with a “1 mil” copper wall has a cross sectional area of $4 \times 10^{-8} \text{ m}^2$. 
Each copper ring removes heat
The surface copper pads and the inner layer pads are much thicker than the plated copper barrel.

The plated copper barrel will **NOT** conduct the heat efficiently.
The plated copper barrel is equal to or greater than the thickness of the copper pads.
Providing Printed Circuit Boards without the Risk

Indianapolis, In
www.divsys.com