

## AN201211-01a On the use of thermal relief pattern for various via

By Denis Lachapelle eng.  
November 2012

### Introduction

There seems to be an eternal discussion about the use of thermal relief on various pad and via types. This application note tries to clarify the use of the thermal relief pattern.

Let start by a definition: A thermal relief pad is a PCB pad connected to a copper pour using a thermal connection. It looks like a normal pad with copper "spokes" connecting it to the surrounding copper.

In the Figure 1 the bottom pattern is a thermal pattern while the top one is an anti-pad, note that a via, or thermal via, would be all black since it is a full connection to the copper pour.



Figure 1

### Why thermal pattern are needed?

If you hand solder a through hole chip ground pin directly connected to ground plane you will discover a problem; soldering becomes very difficult since the soldering iron heat is all sink by the via and ground plane. This problem becomes more serious with heavier copper plane such as two ounce or more, obviously it's also depend on the area of the plane.

To resolve this issue thermal pattern are used in between via barrel and the copper pour; the thermal pattern reduces the total width of copper connected to the copper pour, reducing the thermal conductivity; thus reducing the thermal sink problem.

This problem exists for wave soldering system and some surface mount process.

### Conflicting Requirements

An ideal connection shall have a very low resistance and inductance while thermal relief pattern increases the resistance and inductance. With this knowledge it becomes clear that we shall use them where they are absolutely required and where they have low impact.

## Where to use them?

According to IPC-2221:

**9.1.3 Thermal Relief in Conductor Planes** Thermal relief is only required for holes that are subject to soldering in large conductor areas (ground planes, voltage planes, thermal planes, etc.). Relief is required to reduce soldering dwell time by providing thermal resistance during the soldering process.

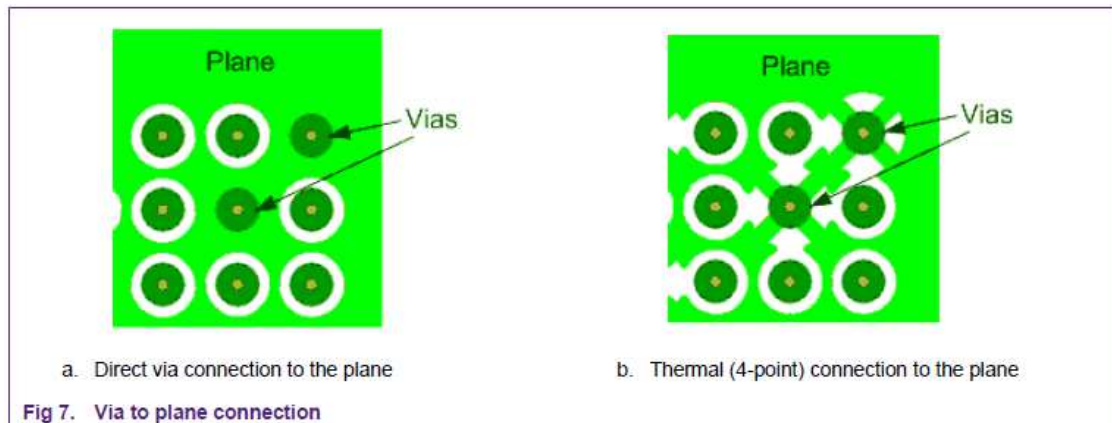
Our views:

- Thermal relief pattern shall be used on power and ground pins of through-hole components or other through-hole pins that are connected to copper pour.
- They shall not be used on surface mount component pad.
- They shall not be used on mounting holes.
- They shall not be used on via.

## BGA Power and Ground

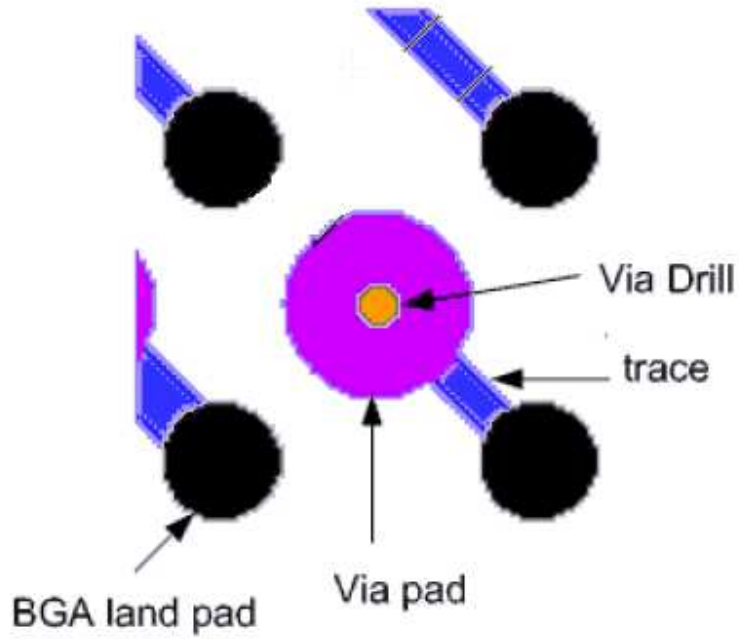
In the previous section we said “thermal relief shall not be used on via” and this is thru for via under BGA connecting to power and ground.

Some EMS recommend thermal relief pattern on via connecting BGA ground or power pins to their respective copper plane. These EMS recommend the thermal relief to equilibrate the thermal sink between signal and power pins; effectively if the thermal sink is much higher for power pins solder joint quality will not be the same. This practice is not recommended since it really breaks the continuity of the ground plane, figure 7, included below, of reference 8 clearly shows this problem. Breaking the ground plane continuity cause the ground plane impedance to increase, this is absolutely not desired.



It is still very important to reduce the thermal sink from the copper pour.

To reduce the thermal sink caused by the copper pour the pad to via trace shall be kept very narrow. NXP in reference 8 recommends 6 mils or less.



## References

1	PCB Layout Considerations for Non-Isolated Switching Power Supplies. Henry J. Zhang, Linear Technology inc., 2012. <a href="http://cds.linear.com/docs/Application%20Note/an136f.pdf">http://cds.linear.com/docs/Application%20Note/an136f.pdf</a>
2	AN-1520 A Guide to Board Layout for Best Thermal Resistance for Exposed Packages. Texas Instruments, 2006. <a href="http://www.ti.com/lit/an/snva183a/snva183a.pdf">http://www.ti.com/lit/an/snva183a/snva183a.pdf</a>
3	Investigation on the efficiency of thermal relief shapes on different Printed Circuit Boards. <a href="http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=5650836">http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=5650836</a>
4	A Demystifying Study of Thermal Relief Pads: Tradeoff between Manufacturing and Cooling <a href="http://ieeexplore.ieee.org/stamp/stamp.jsp?tp=&amp;arnumber=5547325">http://ieeexplore.ieee.org/stamp/stamp.jsp?tp=&amp;arnumber=5547325</a>
5	Impact of PCB via and micro-via structures on component thermal performances <a href="http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=5636350">http://ieeexplore.ieee.org/stamp/stamp.jsp?arnumber=5636350</a>
6	Intel® 840 Chipset Thermal Design Considerations <a href="http://download.intel.com/design/chipsets/applnots/29802702.pdf">http://download.intel.com/design/chipsets/applnots/29802702.pdf</a>
7	Advances in High Speed PCB Design for Layout Professional Scott McMorrow, Teraspeed Consulting Group LLC <a href="http://dcchapters.ipc.org/assets/pnw/presentations/20040915_hsdesign.pdf">http://dcchapters.ipc.org/assets/pnw/presentations/20040915_hsdesign.pdf</a>
8	AN10778, PCB layout guidelines for NXP MCUs in BGA packages NXP Semiconductors <a href="http://www.nxp.com/documents/application_note/AN10778.pdf">http://www.nxp.com/documents/application_note/AN10778.pdf</a>

## Definitions

Thermal via	Thermal vias carry heat away from power devices and are typically used in arrays of about a dozen.
Thermal relief pad	A thermal relief pad is a PCB pad connected to a copper pour using a thermal connection. It looks like a normal pad with copper "spokes" connecting it to the surrounding copper.
Copper pour	The term "copper pour" refers to an area on a printed circuit board filled with copper.
Via barrel	The conductive tube filling the drilled hole forming the via.
BGA	Ball grid array
EMS	Electronic manufacturing services