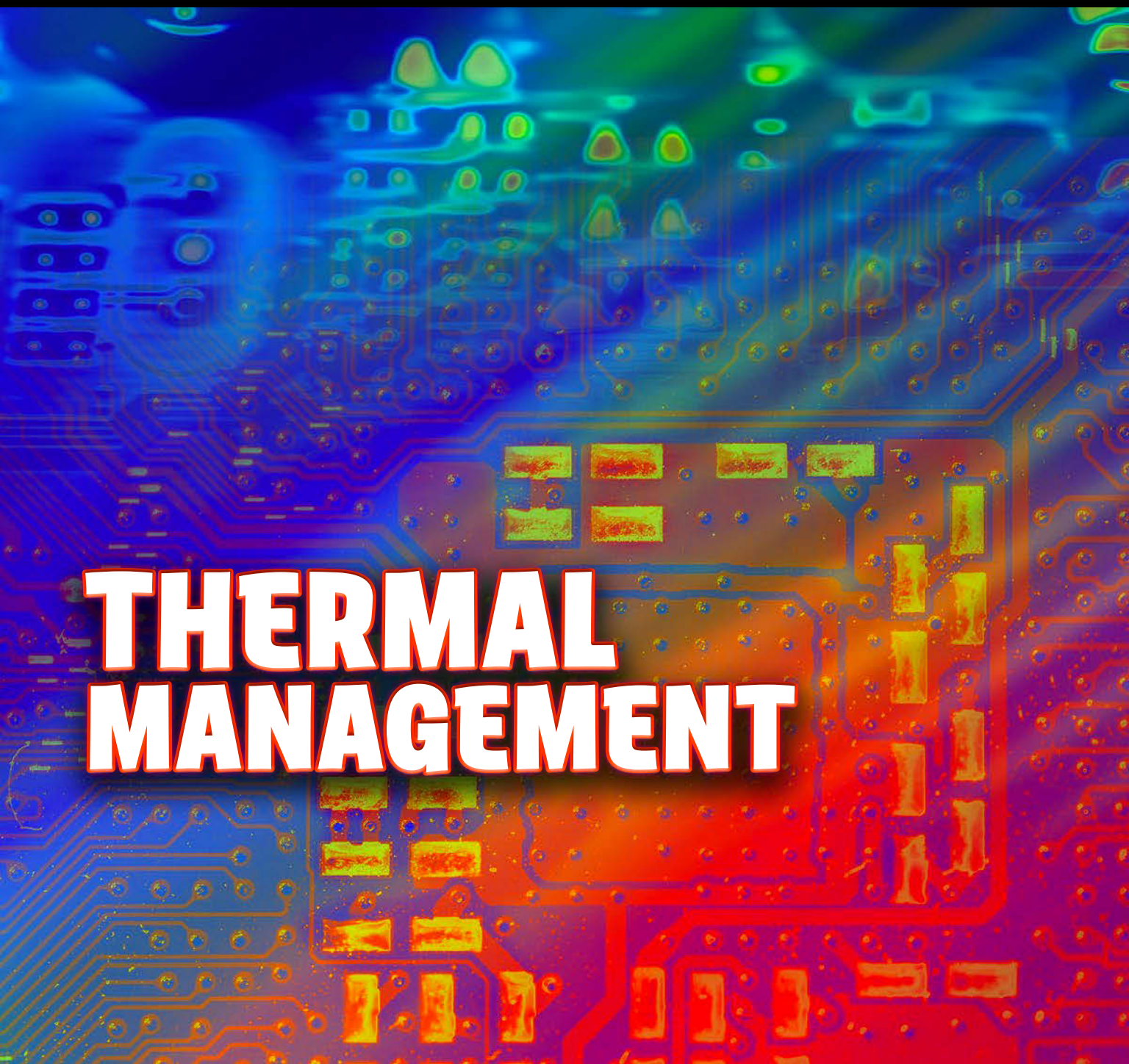
SEPTEMBER 2020

I-Connect007

DESIGN007 MAGAZINE



THERMAL MANAGEMENT

Documenting Your Flex Circuit Design

Flex007 Article by Tony Plemel
FLEXIBLE CIRCUIT TECHNOLOGIES

As a flex circuit applications engineer, when I receive an RFQ, the first thing I do is look at the customer's data and review their manufacturing notes. Quite often, I find notes that supersede IPC specifications in manufacturing documents, as customers often believe these added notes and associated specifications will make the circuit more robust. However, these non-standard IPC manufacturing specifications/notes can wreak havoc on the manufacturing process and can actually lead to a less robust circuit.

For example, a customer will sometimes specify additional copper plating, believing it will result in a more reliable circuit. In reality, that type of requirement can make the circuit less reliable, more difficult to manufacture, and more expensive. When manufacturing yields go down, the price goes up!

In taking a deeper dive into manufacturing notes and the potential issues that they can create, let's use a three-layer multilayer flexible circuit as an example. The first note on a manufacturing print is usually "Manufacture to IPC-6013, Class 2, Type 3." This note should always be included; I cannot stress that enough!

Unfortunately, in the continued review of the documentation, I often find one or more additional conflicting notes further down in the manufacturing notes that overrule IPC-6013 specifications.

Copper average[2,3]	Class 1	Class 2	Class 3
Type 3 and Type 4 except as noted	25 µm [984 µin]	25 µm [984 µin]	25 µm [984 µin]

Table 1: IPC-6013 copper plating requirements.

Copper Plating

One example would be "Minimum copper plating shall be 0.0015"." This note supersedes the IPC-6013 specification in Table 1. PCB designers who are not well-versed in flex circuit manufacturing may not know that exceeding IPC-6013 of 984 µin (0.000984") can cause the circuit to be less reliable and possibly cause problems later in the manufacturing process. Having a specified requirement this large (0.000516" thicker) will require the plating line at the factory to plate more than 0.0015" to ensure the minimum plating is 0.0015" thick.

Also, if there are impedance-controlled traces on the outer layers, we then need POP plating (pads only or button plating). Plating thicker than IPC-6013 recommendations can cause the POP pads to poke through the dry film resist during the roll or vacuum lamination, imaging, and developing process. The exposed bump will then result in pitted etching on the surface of the POP ring, causing rejected parts.

Additionally, this excess plating thickness will then require a thicker cover coat adhesive layer to compensate for the thicker plated bump. The extra adhesive thickness needed to encapsulate the bump will result in more adhesive flow around the pad, causing annular ring issues and also making the circuit more rigid.

On a Type 3 (three layers or more) circuit with 1/2-oz. copper plated to standard IPC recommended panel plating or POP plating, the cover-lay typically only requires 0.002"' of adhesive. However, with the 0.0015" minimum plating thicknesses, the cover-lay may need 0.003" adhesive. Additionally, when

ENIG	Electroless Nickel – Rigid	3.0 – 6.0 µm [118 – 236 µin]	IPC-4552 J-STD-003	b4
	Electroless Nickel – Flex	1.27 – 6.0 µm [50 – 236 µin]		
	Immersion Gold	0.04 – 0.1 µm [1.58 – 3.94 µin][5,9]		b4

Table 2: IPC-6013 ENIG requirements.

laminating FR-4 stiffeners to the flex, the high-pressure lamination process can put undue stress on the plated through-hole barrel, subsequently making the circuit less reliable.

ENIG Final Finish

Another manufacturing note often seen is a customer's own specified ENIG thickness that does not align with IPC-6013 (or IPC-4552) requirements (Table 2). This superseding note can also cause a price increase.

The reason to try to always use the IPC ENIG thicknesses is that the factory sets up their ENIG line to adhere to the IPC ENIG standard. If the customer specifies a different plating parameter that does not meet IPC-6013 specifications, the factory must allow all normal products to completely clear the plating line, then modify the plating line and separately process the non-standard product. The plating line is then changed back to the normal parameters to continue plating the rest of the panels to the standard IPC ENIG parameters. Changing the normally accepted plating specifications unless specifically needed only creates a higher-cost circuit board.

Material Specifications

It is advised not to place detailed material brands on the print unless the customer has specific reasons to do so. Please allow IPC specifications to determine the material selection. For example, it is common for a customer to specify Dupont™ AP8515R (1/2-oz. copper x 1-mil polyimide x 1/2-oz. copper) on the manufacturing print. Simply specifying 1/2-oz. copper x 1-mil polyimide x 1/2-oz. copper on the print, and/or per IPC-4204/11 material is sufficient. This eliminates redundancy, as well as reduces the chances of an error while trying to decipher the manufacturing notes. By calling out materials by name brands, you may have increased the cost of your circuit by 10–20% with no real reason to do so!

Print Templates

Another suggestion is to create print templates and use a fresh template when designing new circuits. If you should have questions on how to lay out a good print template, contact your flex provider and allow them to guide you.

When working with customers, I recommend that they never copy and paste from previous projects, as this leads to copying notes that simply do not pertain to the current project. Using print templates and filling them in on a per-project basis will help eliminate costly mistakes.

If in doubt as to the acceptability of your manufacturing notes, ask your flex engineer to help redline your prints so they will be acceptable to any flex manufacturer. This will eliminate confusing deviations or redlines next time the circuit is built or quoted.

Conclusion

To summarize, utilize IPC-6013 for standard specifications as this allows processes and yields to perform at their best. Keep notes standard and have print templates to work from, and you likely will see improved results related to attaining the products that you desire and an improved cost. In my next article, I will get into details related to certain requirement-driven occasions where one might choose to supersede the IPC-6013 specification, but trust most often in following IPC-6013, and you will be served well! **FLEX007**

Tony Plemel is a senior applications engineer with Flexible Circuit Technologies.