THE pcb design MAGAZINE

April 2015

an I-Connect007 publication

The Future of Nickel in Nickel/Palladium/Gold Final Finishes — p.18

Plus:

Hunter's Two Newest CID Recipients Discuss Certification — p.46

Effective Decoupling Radius — p.52

SURFACE FINISHES

Looking Below the Surface

IPC Plating Sub-committee 4-14: Surface Finish Specifications

by George Milad, page 10

thepcblist.com
thepcblist
THE BEST WAY TO FIND A PCB FABRICATOR, ANYWHERE.

LIGHTNING SPEED LAMINATES

RF Power Capabilities of High-Frequency PCBs

by John Coonrod
ROGERS CORPORATION

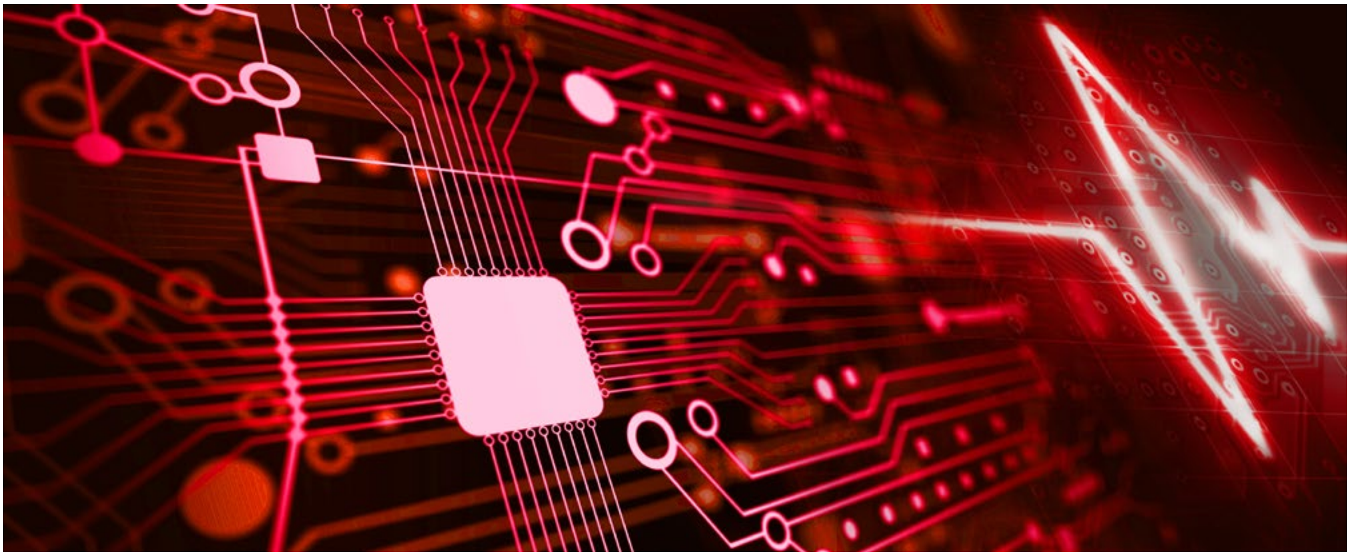
I often hear this question: "How much RF power can be applied to a high-frequency PCB?"

My answer sometimes surprises engineers. I tell them that they can put as much RF power into the PCB as they want, with the assumption that the PCB does not exceed its maximum operating temperature (MOT). MOT refers to the maximum temperature to which a circuit can be exposed without degradation of critical properties. The actual RF power limit of a PCB is based on the MOT of the circuit, and that is dependent on the circuit material, the PCB construction and fabrication process.

The relative thermal index (RTI) is a rating given to UL-rated circuit materials for the maximum temperature to which the raw material can be exposed indefinitely without degradation in material properties. But when the raw material is made into a circuit, MOT is the rating that is most applicable to the power-handling capability of a circuit. The MOT is always less than a circuit material's RTI. When reviewing the maximum RF power-handling capability of a PCB, MOT is used as the maximum temperature of which a circuit can be exposed over long periods of time.

For example, a circuit with a heat rise of +70°C above an ambient of +25°C must endure a temperature of +95°C indefinitely. The RF power which creates this heat rise is acceptable if the circuit has a MOT rating of +105°C. But if the circuit's heat rise is greater than +80°C above ambient, the applied RF power level that created the heat rise would not be acceptable.

When considering circuit heating due to applied RF power, modeling the heat rise of high-frequency PCBs can be difficult. Many variables influence heat rise, and they must be taken into account. Insertion loss is the total RF loss of a high-frequency PCB and is equal to the summation of conductor loss, dielectric loss, radiation loss and leakage loss. Insertion loss is the cause of the heat generated when RF power is applied. A circuit with a high level of insertion loss will generate more heat than a circuit with lower insertion loss, when considering the same



amount of applied RF power. Insertion loss can also be difficult to model, because there are subcomponents that make up insertion loss. Typically, the major contributors to insertion loss are dielectric loss and conductor loss.

Dielectric loss is related to the dissipation factor (Df) and the tangent delta (tanδ) of the material. A material with higher Df causes higher dielectric loss, which in turn can cause higher insertion loss and high temperature rise with applied RF power. Conductor loss is far more complicated than dielectric loss, with several components making up conductor loss. In general, a circuit using copper with a rough surface will have more conductor losses than a circuit using smooth copper. Additionally, there is a circuit thickness relationship, and a thinner circuit will be more prone to conductor loss variables than a thicker circuit. The thicker circuit is more dominated by dielectric loss.

One major consideration for understanding RF power capabilities of a high-frequency PCB is to understand the impact of insertion loss. Generally, a circuit material and design will be chosen to minimize insertion loss, but there are tradeoffs and other issues to consider.

> “One major consideration for understanding RF power capabilities of a high-frequency PCB is to understand the impact of insertion loss. Generally, a circuit material and design will be chosen to minimize insertion loss, but there are tradeoffs and other issues to consider.”

All circuit materials exhibit a property known as thermal conductivity: the measure of the ability to pass heat energy through that material. An extremely good thermal conductor is copper, which has a thermal conductivity value of 400 W/m/K. However, most substrates used for high-frequency PCBs have thermal conductivity values that are considered a thermal insulator or a very poor thermal conductor. Most high-frequency circuit materials have thermal conductivity values in the range of 0.2–0.4 W/m/K. A value of 0.5 W/m/K or higher is considered good for thermal conductivity for a PCB dielectric material.

Now, let’s consider at quick tradeoff. There are some extremely low-loss PTFE materials which can be designed so the circuit will have minimal insertion loss. This means the circuit will generate less heat when RF power is applied and a designer may assume that higher power levels could be applied. However, many PTFE materials have very low thermal conductivity and even though there is less heat generated, the heat cannot efficiently get out of the circuit, so the circuit may heat more than expected.

Another tradeoff to consider is the thickness of the circuit. As an example, a double-sided circuit, which is a simple microstrip circuit bonded to a heat sink, will stay cooler if the circuit is thin, as opposed to thick, when using the same materials and same applied RF power. The thinner circuit has a shorter heat flow path from where the heat is generated at the signal conductor, through the dielectric and to the ground plane below which is attached to the heat sink.

There are several additional tradeoffs to consider, but a quick summary would show that the optimum circuit would use a material with low dielectric loss and smooth copper, which gives low insertion loss and generates less heat. Additionally, the optimum material would have high thermal conductivity and would be relatively thin.

A few high-frequency materials meet these criteria. When working with RF and microwave designs, consulting your materials provider can save you time and money. These companies have plenty of information about thermal conductivity, insertion loss, heat flow, overall thermal management, and much more. **PCBDESIGN**

John Coonrod is a senior market development engineer for Rogers Corporation. To read past columns, or to reach Coonrod, click here.