PCB007 PRESENTS

August 2012

the pcb magazine

AN I-CONNECT007 PUBLICATION

THE FUTURE: Where the ROAD Meets the MAP

A Quick Guide to Next-Generation Materials for Next-Generation Technology

by John Coonrod
ROGERS CORPORATION

SUMMARY: Thermal conductivity, substrate dissipation factor and conductor loss due to copper surface roughness must be considered when exploring the topic of thermal management of PCBs.

Thermal management for PCBs is a complicated topic. In the microwave frequency range, the dissipation factor of the substrate can be important. A substrate with a higher dissipation factor will cause the circuit to have more loss, and loss is associated with heat. A circuit with more loss will generate more heat. Related to loss is conductor loss, which can be aggravated by copper surface roughness. It has been proven that a copper foil with a rougher surface will generate more conductor losses at microwave frequencies; again, more losses and more heat generation. Finally, the thermal conductivity of the substrate can play a very important role in thermal management. An example: If a circuit is a simple microstrip with the signal plane on the top layer and the ground plane on the bottom, it is often the case where the heat is generated near the signal layer and the ground plane will have a heat sink attached. The heat flow path will need to migrate through the substrate to the ground plane and heat sink in order to have improved thermal management. A substrate with increased thermal conductivity improves the heat flow path in the substrate.

Most PCB substrates have a thermal conductivity property with a value around 0.3 W/m/K. Some of the ceramic-filled substrates are much higher and are around 0.6 W/m/K.

There are multiple reasons for a substrate to have a very low dielectric constant. Currently the lowest dielectric constant (Dk) materials on the market are typically PTFE-based and have a Dk around 2.2 to 2.5. One obvious benefit to the lower Dk is that to maintain a controlled impedance circuit, with a set construction, the conductor width will be wider when using a substrate with the lower Dk. When there is a need for very thin substrates it is sometimes difficult to have a conductor wide enough to be robust in PCB manufacturing and still achieve the impedance target. The lower Dk substrate will allow a wider conductor and in Table 1 there is a simple example of the different conductor widths, when using substrates of different Dk values.

The move toward halogen-free has been critical in some areas of the world and some industries; however, the migration to halogen-free materials has been much slower in others. There are several challenges to developing a halogen free substrate, some of which are peel strength, good flame-retardant properties, low dissipation factor, thermal robustness and good characteristics for general circuit fabrication process needs.

Laminate Thickness (mils)	Material dk	Conductor Width (mils)
2	4.5	3.3
2	2.2	5.6
3	10	1.5
3	4.5	5.1
3	2.2	8.6
4	4.5	7
4	2.2	11.7
5	4.5	8.9
5	2.2	14.8

Table 1: Conductor width variation due to substrate thickness and Dk, while using a microstrip transmission line circuit.

Throughout the years, there has been a tremendous amount learned regarding changes due to aging in substrate properties. Any realist will tell you that everything changes with time and of course that is true with all PCB materials.

A related topic in the microwave industry that has received ample attention during the past several years is copper surface roughness. It has been found that the surface roughness of a conductor will impact the total insertion loss of a microwave circuit. Furthermore, it has been found that the surface roughness will also affect the wave propagation constant which has an effect on the apparent Dk, which is the dielectric constant of the laminate that would be implied by the performance of the circuit under evaluation. In some cases the copper surface roughness can make a circuit behave as if it were on a laminate with a Dk that is different than the actual measured Dk of the laminate.

Additional information about this topic can be found on the Rogers Technology Support Hub located on the Rogers website: rogerscorp.com. **PCB**

John Coonrod is a market development engineer for Rogers Corporation, Advanced Circuit Materials Division. About half of his 25 years of professional experience has been spent in the flexible PCB industry doing circuit design, applications, processing and materials engineering. Coonrod has also supported the high-frequency, rigid PCB materials made by Rogers for the past 10 years. Reach Coonrod at john.coonrod@rogerscorporation.com.

In-Circuit Design Presents Stackup Tech Webinar Series

In-Circuit Design's Stackup Technology Webinar Series focuses on design tradeoffs designers are forced to make when planning a multilayer PCB for today's high-speed, digital designs.

With inherent signal integrity and power delivery requirements, today's PCBs require far more discipline in material selection and the arrangement of layers in the stackup. The objective of this webinar series is to guide design teams—from hardware engineers to layout designers and fabricators—through the process of evaluating and selecting the right laminate for any design, creating PCB stackups that meet the requirements of complex, multilayer boards that work right the first time.

Who should attend:

- Hardware engineers and PCB designers interested in getting it right the first time
- Engineering and CAD managers, who want to improve quality, reliability, schedule, and cost control in the R&D process
- EMC and test engineers desiring to push proactive design practices upstream in the design process
- PCB fabricators desiring to enhance collaboration, communication, and quality with product teams that they support

The remaining Webinar Sessions/Topics:

- August 15—Material Selection for Multi-Gigabit SERDES Design
- August 22—Signal-Layer Selection: Microstrip, Buried Microstrip, Single and Dual Striplines
- August 29—Selecting a PCB Fabricator
- September 5—Decoupling/Bypass Capacitor Selection

Select this link to register, and choose between the 9 a.m. or 5 p.m. PST sessions. Once registered, you will receive e-mail instructions for logging into each webinar. All webinar sessions will be less than one hour. For more information, click here.