

THE **pcb**  
**DESIGN**  
MAGAZINE

June 2014

**Flexible Circuitry:  
a 3D Packaging Tool**  
by Dave Becker  
p.24

**Ninja Flex Circuits**  
by Tom Woznicki  
p.34

AN  I-CONNECT  PUBLICATION

## **Designing Flex Circuits**

# **Why Procuring Flexible Printed Circuitry is Different**

by Thomas Stearns, page 12



[thepcblist.com](http://thepcblist.com)

**the pcblist**  
THE BEST WAY TO FIND A  
PCB FABRICATOR,  
ANYWHERE.

# When is Controlled Impedance Important?

by John Coonrod

ROGERS CORPORATION, ADVANCED CIRCUIT MATERIALS DIVISION

The importance of controlled impedance hinges upon many variables, such as the PCB's characteristics and how it is to be used. A PCB designed for digital applications will often have different impedance requirements than a circuit designed for RF applications. Within both of these categories, however, there are sub-categories of specific types of applications.

Digital applications, especially high-speed digital applications, will require consistent and controlled impedance values for signal integrity purposes. There are many different methods for checking a PCB for good signal integrity: eye diagrams, pulse distortion, bit error rate, and skew. A critical trace on a digital board with impedance variation can impact these quality measurements of the PCB.

Typically, when energy of a varying signal goes from one impedance environment to another, there will be some amount of reflection in the transition. A digital pulse going from one area of a PCB with 40 ohms impedance to another area with 50 ohms will have some reflected energy at that transition. Along with that, the pulse amplitude and shape may vary as well, depending on the significance of the reflection. The pulse distortion will impact the previously mentioned signal integrity concerns for a high-speed digital system.

Another issue with an unmatched impedance area of a PCB for digital systems is electromagnetic

interference (EMI). The reflection associated with the impedance mismatch will cause electromagnetic radiation in the localized area of the transition. That radiation can couple its energy to neighboring traces or components on the board, causing electrical performance distortion of those items.

Impedance matching for RF applications sometimes share similar issues associated with high-speed digital applications. Many times in RF applications, there is a need for efficient energy transfer from one module to another. A simple example would be to consider the energy generated within the transmitter portion of a radio and how to transmit that energy to the antenna efficiently. If the feed line from the transmitter to the antenna is not well matched for impedance, some energy will be lost before it can be transmitted. The transmitting portion of the radio will not function as well as it should, possibly shrinking the distance at which the signal can be received decreasing the clarity of the signal received.

There is a large study within the RF/microwave and millimeter-wave design community devoted solely to impedance matching. Besides the simple radio transmitting example given, there are other issues with power amplifiers, radar, low noise amplifiers (LNA), and many more. Within each of these applications there may be multiple areas where impedance matching is critical.

In the case of power amplifier circuits, the PCB typically has many different areas of functional-



**WHEN IS CONTROLLED IMPEDANCE IMPORTANT?** *continues*

ity. There are also multiple areas of impedance matching, and the impedance values can purposely change greatly from one area to another on the circuit. The power amplifier IC itself will often have input impedance of much less than 10 ohms, whereas the traces on the PCB will be 50 ohms. To get the energy to efficiently transition from 50 ohms down to less than 10 ohms, it is critical to have clean signal energy input into the power amplifier IC.

There are several different types of impedance. The most common type of controlled impedance on a PCB is referred to as characteristic impedance. Other types include input impedance, wave impedance, image impedance, and others. Many of the impedance issues are frequency-dependent, and this can be an issue for wideband power amplifiers. The impedance matching network that is necessary for the design of a power amplifier can only tune the impedance transitions over a range of frequencies. Many times the actual power amplifier IC can be used over a wider range of frequencies (wide-

band) than the tuning networks that are used to match the impedance for traces going into and out of the chip.

The impedance value of a PCB is dependent on many variables. In order of their significant impact on impedance, these variables include: thickness of the substrate, dielectric constant, conductor width and copper thickness. High-frequency circuit materials are formulated to have a tight thickness control as well as dielectric constant control. The same can be said for the laminate's copper thickness. But remember that this thickness can vary on the final circuit because of differences in PCB fabrication techniques. **PCBDESIGN**



John Coonrod is a market development engineer for Rogers Corporation, Advanced Circuit Materials Division. To reach Coonrod, [click here](#).

**video interview****PCB Auditing at the Next Level**

*by Real Time with...*  
**IPC APEX EXPO**



What capabilities should you look for when you're selecting a new fabricator? Is a survey good enough, or should you require a full audit? Rob Scott of the auditing company Next Level PCB explains the criteria that he assesses when auditing a PCB supplier.



**realtimewith.com**

