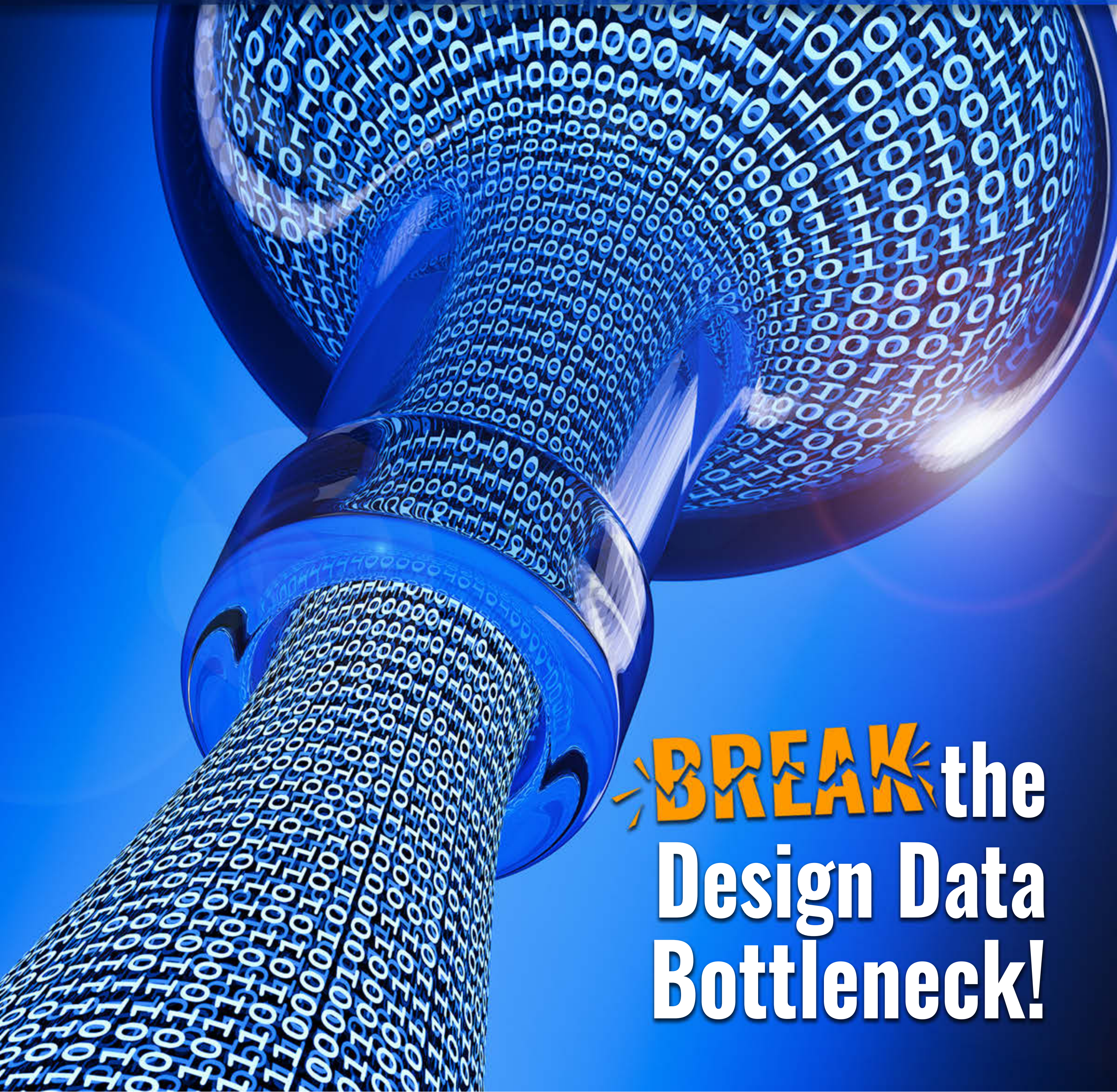
I-Connect007

OCTOBER 2018

DESIGN007 MAGAZINE



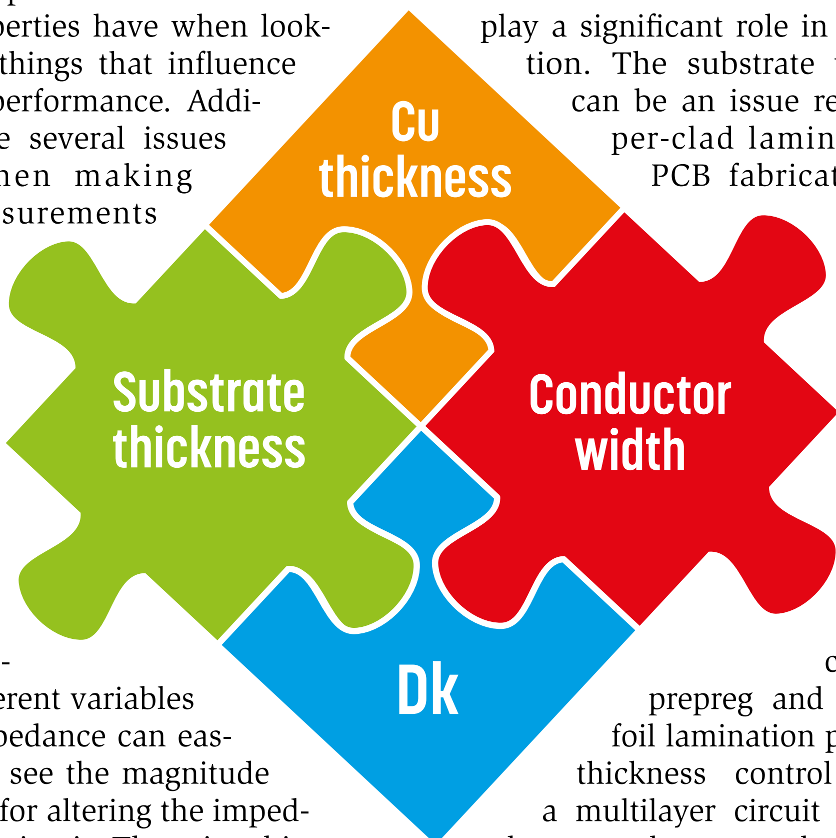
BREAK the Design Data Bottleneck!

Different Aspects of Impedance for PCBs

Lightning Speed Laminates

by John Coonrod, ROGERS CORPORATION

Many PCBs are specified to have a controlled impedance value. There are several circuit and material properties that impact the impedance performance of a circuit. Some of these properties are obvious to engineers who have worked with controlled impedance boards over the years. However, even experienced engineers are sometimes surprised to find the level of impact these properties have when looking at all of the things that influence PCB impedance performance. Additionally, there are several issues to consider when making impedance measurements to ensure the values are accurately captured.



As a reference, consider a microstrip transmission line circuit. When an impedance model is generated for a PCB microstrip construction, the different variables that influence impedance can easily be changed to see the magnitude each variable has for altering the impedance value of the circuit. There is a hierarchy of the influences of these variables on PCB impedance. For a microstrip transmission-line circuit, the most to least influential variables include substrate thickness, conductor width, copper thickness, and dielectric constant (Dk).

As a material supplier, we are often asked to investigate the Dk of our material when a circuit has an issue with incorrect impedance. In reality, the Dk is the least impactful variable for impedance. It is much more common for substrate thickness and conductor width to play a significant role in impedance variation. The substrate thickness concern can be an issue related to the copper-clad laminate and/or the PCB fabricator, depending on how the circuit construction. In general, the substrate thickness control is better for a microstrip circuit using the copper-clad laminate compared to a microstrip that a PCB fabricator might construct using a prepreg and copper foil for a foil lamination process. Due to the thickness control issue—assuming a multilayer circuit with a microstrip layer on the top or bottom of the structure—a circuit with a demanding impedance specification is usually best when made with a laminate from the material supplier as opposed to a foil lamination at the PCB fabricator.

Controlling the signal conductor width is more critical for circuits that use thin substrates because thinner substrates usually have narrower conductors. A normal conductor width tolerance is ± .5 mil, but this can depend on the copper thickness and circuit density. For a circuit using a thick substrate and a wide signal conductor, the common conductor width tolerance has less influence on impedance variation than a thin circuit with a narrow conductor. The impedance of a circuit using a narrower conductor is impacted more by the conductor width tolerance than a circuit using a wider conductor.

The impedance of a circuit using a narrower conductor is impacted more by the conductor width tolerance than a circuit using a wider conductor.

Copper plating is applied in the process to generate plated through-hole vias and has some thickness variation. Even though the copper thickness variation can impact the impedance performance of the circuit to a lesser degree than substrate thickness and conductor width, a larger concern is usually how thicker copper can increase the conductor width tolerance. Thicker copper cannot be etched as accurately for good conductor width control as thinner copper; however, there are exceptions, depending on the image and etching processes.

The most influential variables should be considered for impedance control. If the PCB design has a tight specification for impedance, a substrate with strong thickness control should be considered along with a well-controlled process for generating the signal conductor width. Copper thickness control and Dk control are important as well, but these are typically less critical for impedance accuracy. However, Dk consistency is extremely influential for the phase response of the radio-frequency (RF) circuit.

Additionally, the measurement of impedance has many issues to consider. Typically, a time-domain reflectometer (TDR) is used to measure the impedance of a PCB. There are many types of TDRs with different capabilities. In general, the rise time of the TDR is essential and a faster rise time will allow more accurate impedance measurements. Further, there can be an issue with masking, which is a term used to explain that a large impedance anomaly can mask the true impedance anomaly behind it.

The basic operation of a TDR is to send a pulse down a circuit and look at the reflected energy to determine the impedance. It is not uncommon to have a large impedance anomaly where the test probe meets the circuit being tested for impedance. This impedance anomaly will reflect a lot of energy back to the TDR, and less energy will propagate down the circuit. This makes other impedance measurements after the impedance anomaly somewhat altered.

It is relatively easy to evaluate this issue by testing a circuit that has a good impedance anomaly where the probe meets the circuit, then use a tool to alter the circuit pattern in the area where the probe touches the circuit and causes a worse impedance anomaly. After this, retest the circuit and a change in impedance will often be detected. Depending on the TDR's capabilities and the circuit construction, the impedance of the trace will change from the original measurement. In some cases, it may be very little, and in others, more significant. **DESIGN007**

John Coonrod is technical marketing manager at Rogers Corporation. To read past columns or contact Coonrod, click here.