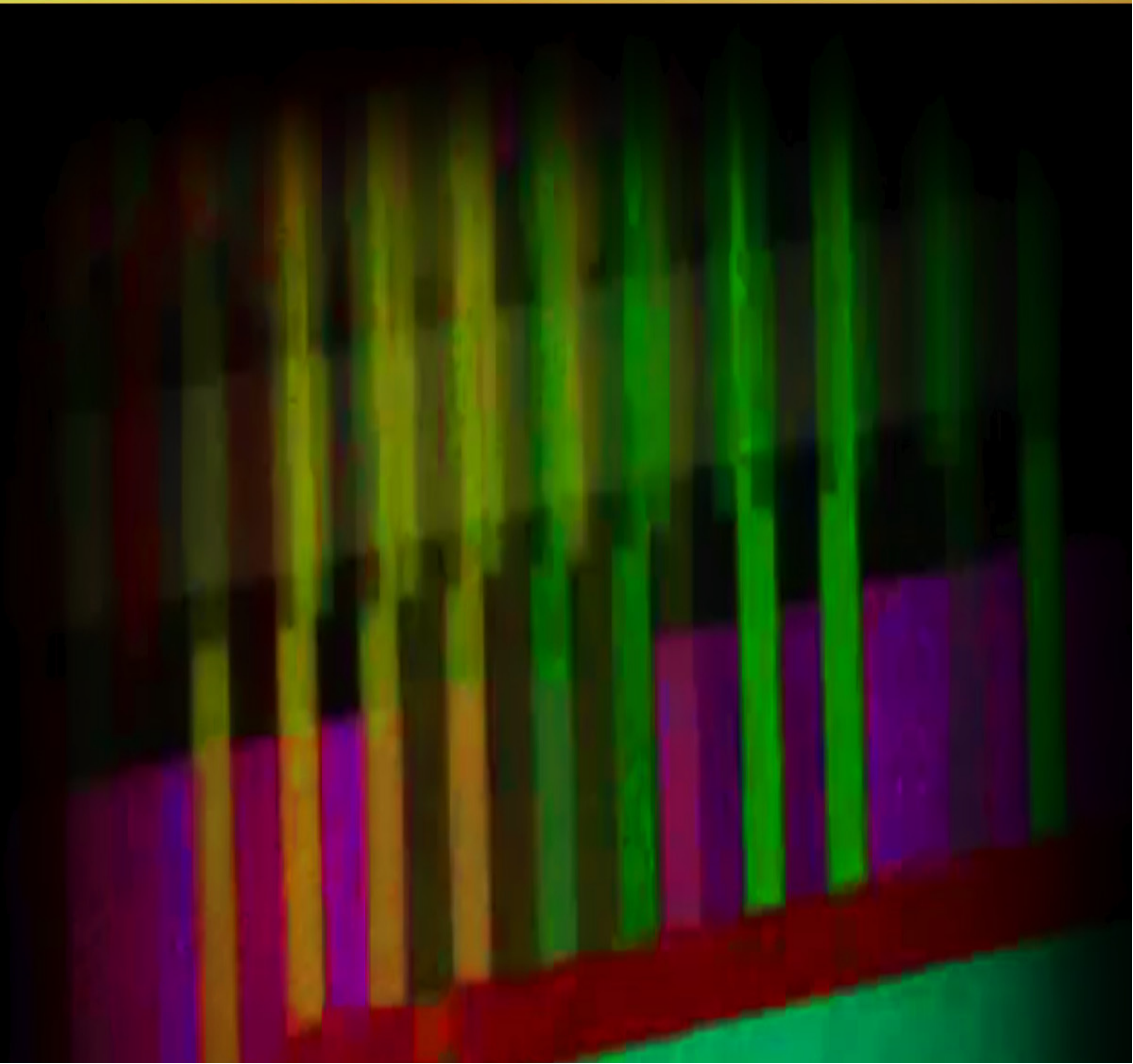
# THE PCB DESIGN MAGAZINE

December 2012

AN I-CONNECT007 PUBLICATION



## ROUTING AND PLACEMENT ISSUE

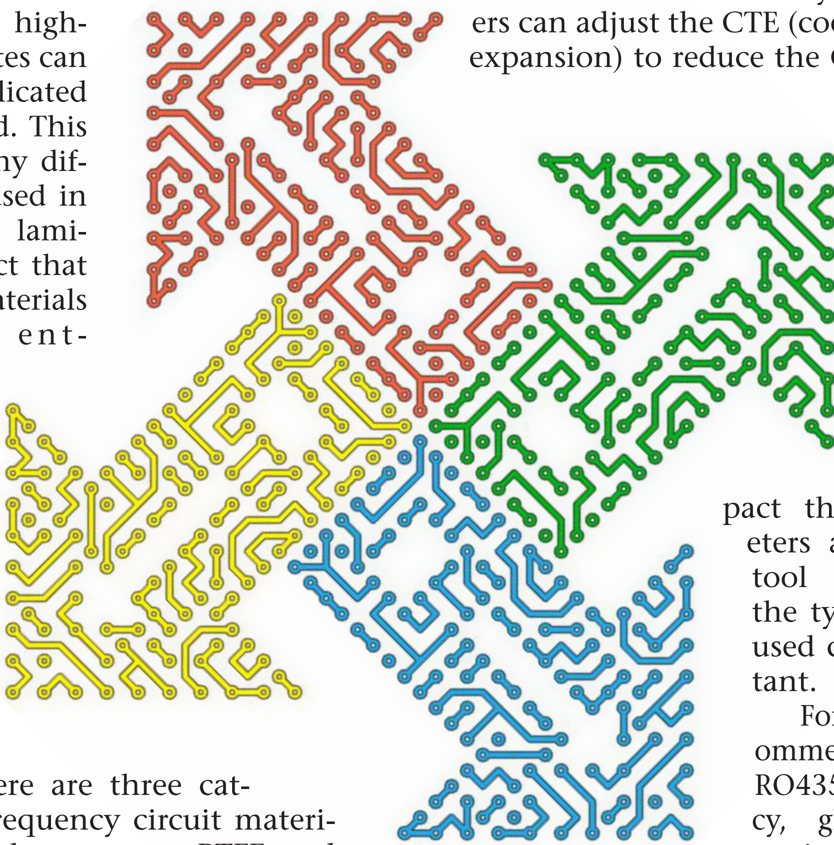


# The Challenge of Routing High-Frequency Laminate PCBs

**by John Coonrod**
ROGERS CORPORATION

***SUMMARY: Yes, we're talking about the other type of routing. Designers of high-speed PCBs favor high-frequency laminates for their low loss and tight impedance control, but these materials can cause trouble for the router operator.***

The routing of circuits made from high-frequency laminates can be more complicated than first assumed. This is due to the many different materials used in high-frequency laminates, and the fact that some of these materials will route differently than others. Another point: These materials are sometimes used as a hybrid construction, meaning dissimilar materials are combined to form the PCB.



Generally, there are three categories of high-frequency circuit materials: filled hydrocarbon systems, PTFE, and filled-PTFE systems. Within each of these families of products there are also glass woven reinforced and non-glass woven reinforced substrates. Additionally the filled PTFE systems often use different fillers in order to achieve certain electrical properties. All of these combinations of materials, fillers and reinforcement can have an impact on the cut-edge quality when routing PCBs.

The filled hydrocarbon systems are typically ceramic-filled. The many types of ceramic fillers have varying attributes. Electrical properties and thermal characteristics must be considered when choosing a filler type. The ceramic filler is used to adjust the dielectric constant and will add to its stability. Many ceramic fillers can adjust the CTE (coefficient of thermal expansion) to reduce the CTE to be closer to that of copper. Some ceramic fillers will be more or less abrasive to the drill and routing tools. In the case of the ceramic-filled hydrocarbon laminates, the filler will impact the routing parameters and especially the tool life. Additionally, the type of routing tool used can be very important.

For instance, the recommended router for RO4350B, a high-frequency, glass-reinforced, ceramic-filled, hydrocarbon-based laminate, is a carbide multi-fluted spiral chip breaker or a diamond cut router bit. It is recommended to have the copper etched away in the router path. Some general routing parameters are shown in Figure 1.

These recommendations are generic and for illustration purposes. However, it is suggested that the material supplier should be

| | |
|---|---|
| Surface speed | <500 SFM |
| Chip load | 1.0 to 1.5 mil/rev |
| Tool life | >30 linear feet |

**Figure 1:** General routing parameters for certain high-frequency laminates.

involved in determining the proper routing conditions for a PCB using its materials.

The non-glass woven, ceramic-filled hydrocarbon materials are typically used in niche applications and have similar routing concerns. These materials are brittle and can fracture during different stages within the PCB fabrication process. However, routing is typically not a concern for fracturing. These materials are offered with a variety of dielectric constant and the filler used to adjust the dielectric constant has a different level of abrasion characteristics. Typically, the materials with the lower dielectric constant (4 or less) will be more abrasive and decrease the tool life. Furthermore the abrasive nature will cause the material to be more susceptible to poor edge quality issues such as burring. Again, the material supplier should be consulted for the optimum routing conditions for circuits made with its materials.

The PTFE-based high-frequency substrates often have different concerns than ceramic-filled hydrocarbon materials. The PTFE materials are relatively soft and can easily smear and have stringers at the routed edge. There are different techniques used to minimize this concern. A slower surface speed is typically used to reduce heating and a double-pass routing pattern is used as well. The double-pass routing is done in two different directions; the first pass may be counterclockwise, whereas the second pass would be clockwise.

The ceramic-filled PTFE systems are more forgiving in the PCB fabrication process than the non-ceramic-filled PTFE substrates. The addition of the ceramic filler adds some rigidity to the substrate as well as raising the thermal conductivity. The raise in thermal conductivity may slightly reduce the issue of overheating the substrate and causing smear during routing, however it still can be a concern. Sometimes this material is also available with woven glass reinforcement and the routing concerns are similar. However, the reinforced substrate may have slightly more issues with edge-cut quality. When routing either the reinforced or non-reinforced PTFE substrates and minimizing debris is critical, it is often recommended to have pre-routed vacuum channels in the backer boards.

Hybrid PCBs are becoming increasingly common. The combination of dissimilar materials used to create a multilayer PCB has many advantages, but there can be several fabrication-related issues and one of these may be routing the circuits. When the dissimilar materials are significantly different in modulus, there is sometimes a concern at the interface of these materials regarding routing. Basically, the concern is that routing issues become complicated at the transition from soft to hard materials. The soft material will want to stretch and cause stringers, while the rigid material will want to cut away clean. These material transitions can be challenging depending on thickness differences and where the transitions occur in the circuit stack-up.

In general, the routing parameters should trend toward the needs of the soft materials, and as previously mentioned, the material supplier should be involved with the routing concerns to offer assistance in optimizing this process. **PCBDESIGN**

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.