

Design Notes for Laminates and Other Substrate Materials

By Gary Breed
Editorial Director

Here are a number of manufacturers' recommendations and other notes on design, handling and processing of circuits on common substrate materials

The final realization of a circuit design is its construction on a substrate of some sort. The most common substrate for high frequency circuits is a printed circuit board, which

may range from a low cost fiberglass laminate such as FR-4 to an advanced low dielectric constant glass reinforced PTFE or ceramic/hydrocarbon laminates intended specifically for microwave circuits.

Using these laminates properly is important. Design engineers are certainly concerned with characteristics that directly affect electrical performance, including dielectric constant, dissipation factor and copper thickness (resistivity). A designer also needs to know how these factors may change with frequency, as well as with temperature and humidity. Often, the circuit designer is not concerned with mechanical factors, leaving those to the board fabricating contractor. However, like many other aspects of design, understanding more than the minimum necessary facts helps avoid surprises. This tutorial provides an introduction to mechanical and manufacturing related factors. To do this we'll draw from the laminate manufacturers' own guidelines.

Rogers Corporation RO4000® Microwave Laminate

The data sheet for Rogers RO4000 [1] includes detailed guidelines for processing. This laminate uses a glass reinforced hydrocarbon/ceramic dielectric to obtain a low dielectric constant, low loss and stable electrical properties. To maintain these high perfor-

mance electrical properties during manufacturing, specific processing recommendations should be followed. An abbreviated list of topics covered by these recommendations includes:

- Drilling tool sizes, forces and speeds
- Deburring instructions
- Copper plating of holes and vias
- Resist and etching processes
- Mechanical routing parameters

It is clear that proper mechanical processing of this material is a high priority for board production, whether performed by a contract supplier or in-house. A design should be especially concerned about the consistency between prototype fabrication and final production, since these may be performed in different facilities.

PTFE Laminates: Recommendations from Taconic Advanced Dielectric Division

While much attention is given to the actual processing of circuits using PTFE dielectric laminates, the physical properties of the material require additional knowledge of handling the material during the manufacturing process. The performance obtained with careful design, layout, etching and assembly can be compromised with improper handling. Some guidelines offered by Taconic [2] are:

- Do not mechanically scrub or abrade the material. When the copper is removed, PTFE has an excellent surface for bonding adhesives, solder mask, etc. Any mechanical cleaning or roughening process adds risk of deforming the material

or pressing contaminants into the soft PTFE surface.

- Avoid bending—do not pick up material by a corner or end. This may stretch the copper and deform the board.
- Prevent contaminant deposits by handling with gloves and using a slip sheet between stacked boards. This helps avoid embedding unwanted particles in the material. When needed, use chemical cleaning.
- Minimize the need for deburring by carefully controlling drill operation. Deburring, like other mechanical operations, can damage the material.

These recommendations make perfect sense for the relatively soft PTFE dielectric material. Even when reinforced with glass fibers, these laminates are more prone to mechanical damage than other materials.

Thermal Management on Laminates: Arlon Electronic Materials

Smaller size products and greater functionality can result in higher heat concentrations than experienced in earlier generations of a product. One method to deal with this issue is to use laminate materials with enhanced thermal conduction properties. Arlon has developed thermally conductive PTFE and FR-4 laminates that greatly increase the ability of the printed circuit board to remove heat from the electronic components.

Reliability of semiconductors follows a pattern of 10°C increase in junction temperature doubling the failure rate (half the mean time to failure). This demonstrates the value of all thermal management techniques. Existing techniques for board-level control include heat sinks, thermally conductive adhesives and fillers, metal backplanes

and thermal vias. External cooling with fans, heat pipes and air conditioned operating space are also commonly employed. Each of these techniques has its advantages and disadvantages in cost and/or performance.

The development of thermally conductive laminates provides another option for enhanced thermal performance. Besides removing heat from power devices, a stable thermal environment improves reliability by reducing the magnitude of expansion and contraction.

Thermal performance of circuit board materials is not a typical part of design, but the option is available for designs that can benefit from its performance contribution.

FR-4 Design and Manufacturing Guidelines: Plextek Ltd.

For lowest cost, RF products may use the higher dielectric constant of low-cost, easy-to-process FR-4 laminates. To help educate design engineers in the proper use of FR-4 materials, Plextek Ltd., a contract manufacturer, provides the following guidelines [4]:

- Know PC board fabrication limitations. Minimum track thickness of FR-4 is typically 0.2 mm, tolerance of line width is ± 0.025 mm, and for lowest cost, via holes should be no smaller than 0.5 mm, although they can be smaller if necessary.
- Understand parasitic capacitances and inductances that result from component mounting on a PCB. Traces are inductors, pads are capacitors, so their dimensions will determine the magnitude of these additional reactances. Some component manufacturers have characterized their devices on typical PCBs, which can help designers accurately incorporate parasitics into the design.
- Ground planes are a necessity for RF circuits, and there are

many options for their implementation. For example, separating digital and analog grounds is a common practice, but it is also possible that a single low-impedance ground plane will provide better overall isolation and reduce crosstalk.

Other Substrate Materials

In addition to classic copper-clad laminates, high frequency circuits may be assembled on many other substrates.

LTCC (low temperature co-fired ceramic)—LTCC is common for multi-element “components” such as mixers, couplers, filters, amplifiers and antennas. Although LTCC construction has some similarity to multilayer PC boards, they offer the lower loss of ceramic material and support printed thick-film passive components such as resistors and inductors. LTCC supports smaller geometries than PCBs as well.

Microwave “hard” ceramic—Alumina ceramic substrates are a traditional way to build high performance microwave circuits. The dielectric constant is tightly controlled, mechanical properties are highly stable, and thick film processes work well for lines and passive components. Give its long time use, there is a well-established knowledge base for this material.

Semiconductor and semi-insulating materials—Today’s System-on-Chip (SoC) and related module technologies are based on semiconductor processing rather than the mechanical and chemical processing of PCBs. The imperfect dielectric properties of these materials results in greater design and processing challenges than older techniques.

Although there are other substrate choices, they are much less common than the ones chosen for this article.

References

1. “RO4000® Series High Frequency Circuit Material,” Data Sheet, Rogers Corp., Advanced Circuit Materials Div., www.rogerscorporation.com
2. “The Care and Handling of PTFE Laminates,” Technical Note, Taconic Advanced Dielectric Div., www.taconic-add.com
3. Russell Hornung, “Expanding the Thermal Management Tools for RF Infrastructure & Power Amplifiers,” IWPC 2007 presentation, Arlon Electronic Materials, www.arlon-med.com
4. Geoff Smithson, “Practical RF Printed Circuit Board Design,” Plextek Ltd. technical note, www.plextek.co.uk