High Frequency Circuit Materials used in the PCB Industry

John Coonrod, Rogers Corporation

Specialty high frequency circuit materials have been used in the PCB industry for decades and for many different reasons. There are several attributes of these materials that are very unique when comparing to the more traditional PCB materials. When these attributes are well understood, the PCB fabricator and the OEM can benefit greatly from improved electrical performance. And there are many other non-electrical improvements that can be achieved of which the general public in the PCB industry may not be aware. In order to realize the full potential of the benefits these materials offer, the PCB fabrication issues must be well understood.

Most high frequency applications will have multiple demands for the PCB materials and not just one specific item of interest. For a specific application, many times one concern is paramount while the others are secondary. Understanding these needs and the attributes of the high frequency materials, will allow a person to choose the optimum material for the application. A couple of example applications will demonstrate the various attributes to consider.

A simple application example could be a small filter circuit using high frequency materials that will be soldered to a larger FR4 circuit board, as shown in figure 1.

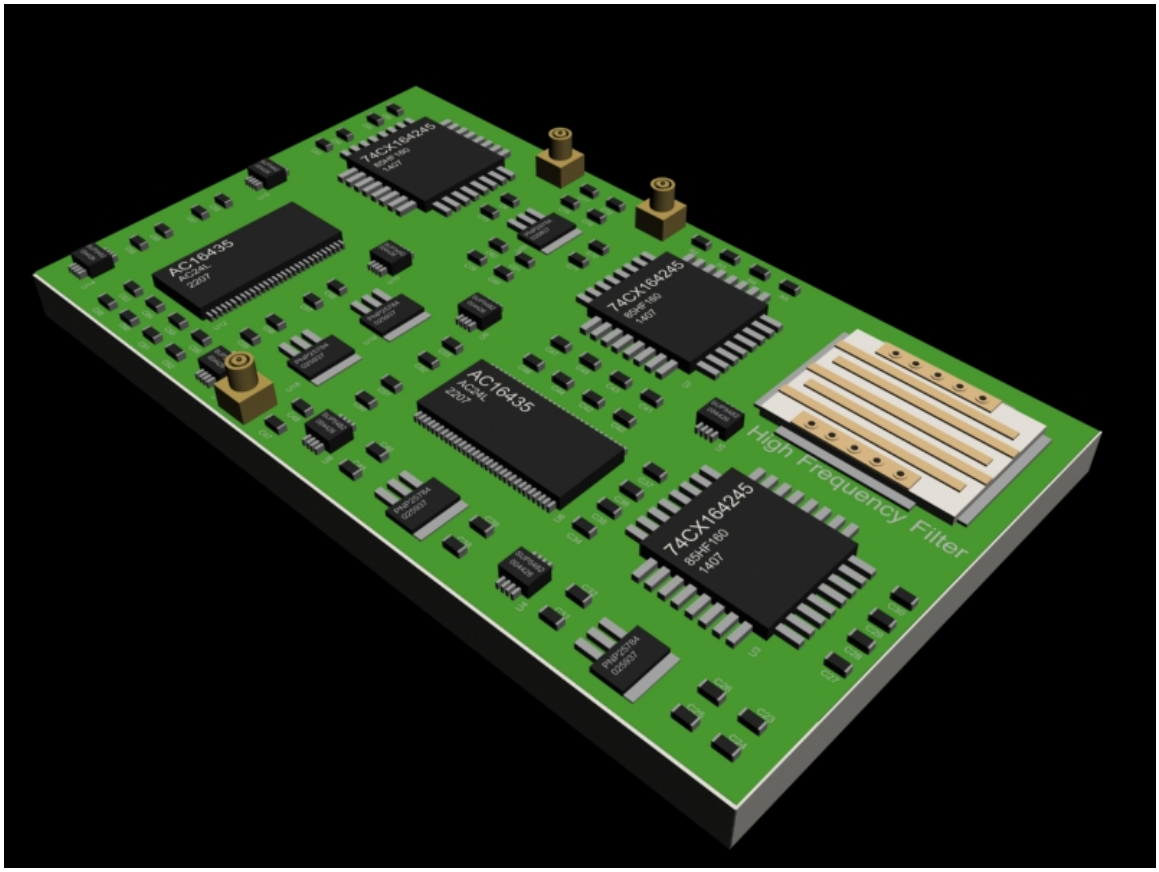


Figure 1. Example of a PCB, with a filter soldered to it.

The completed assembly will be encased inside a sealed enclosure and will operate in an environment that will vary greatly due to outdoor seasons and equipment heat generation. Temperatures inside the enclosure could vary from –25C to +70C.

Filter circuits will typically need a material with very consistent dielectric constant (dk). This means consistent dk within a sheet of circuit material as well as from lot-to-lot. This is required so the filters being attached to the

FR4 circuits will have the same performance and minimal tuning will be necessary. This filter will be a double-sided (2 copper layer) plated through-hole (PTH) circuit, with a ground plane on the bottom and the signal plane on top; microstrip edge coupled filter. As previously mentioned, most high frequency applications have more than one demand on these circuit materials. In this case the consistent dk is paramount, but there are other demands that are implied and not specified yet.

All circuit materials have some amount of growth / shrinkage due to heating / cooling of the material. That is due to the coefficient of thermal expansion (CTE) of the material. In this case the CTE may not appear to be important, but it can be critical. Once the filter is soldered in place to the FR4, the solder joints are a rigid bridge between the FR4 circuit and the high frequency filter. When the unit experiences a change from 0°C to +30°C (as an example), then the FR4 and the filter circuit will want to expand. If the CTE is significantly different between the FR4 and the filter, then stresses can develop at the solder interface, which connects them. This may be an issue over time, where one excursion from 0°C to +30°C is not a problem, however with repeated stress exercising the solder joints over time then the soldered connection could work harden, increase resistance and cause electrical anomalies.

Another possible critical issue that may not be apparent for this application is the TCdk of the circuit material. Each type of circuit material has a property where the dk value can change with a temperature change and that is the thermal coefficient of dielectric constant or TCdk. So with a certain circuit material the dk may appear to be good and consistent when tested at room temperature, however in the application and as the temperature changes then the dk can change and cause the filter to perform significantly different.

An additional consideration could be the assembly operation, where the filter is soldered to the FR4 circuit board. Since the filter has plated through-hole via's, then the z-axis (thickness) CTE could be a concern during the soldering operation. Depending on the design of the filter and / or the FR4 circuit, there may be a need to dwell at elevated soldering temperatures for a longer period of time. If the circuit material has a higher z-axis CTE, then the dwell at elevated temperatures can cause damage to the plated through-hole via's and degrade the electrical connection from the signal plane to the ground plane.

Lastly if the filter is intended to operate over a wide range of frequencies, most circuit materials will have different dk values at different frequencies. Some materials are more stable than others, however this may also need to be considered.

In summary for this application example, this filter circuit should use a material that has a tight tolerance for dk, a CTE that is closely matched to FR4 in the x-y plane, a low TCdk, a low z-axis CTE and a stable dk vs. frequency curve. There are many high frequency circuit materials to choose from, however finding the right circuit material, which fills all of these requirements, will reduce the available choices. A comparison between a standard FR4 circuit material and a good choice for this application is shown in figure 2.

	dk	dk tolerance	x-y axis CTE (ppm/C)	z-axis CTE (ppm/C)	TCdk (ppm/C)
R04350B™	3.66	+/- 0.05	15	35	50
Standard FR4	4.5	+/- 0.30	14	65	300

Figure 2. Comparison between Rogers R04350B™ and a standard FR4 material

Additionally, if it was critical to reduce the physical size of the filter then a different circuit material may have been chosen with a high dielectric constant. A material with high dielectric constant will allow the circuit to be reduced in size and still perform the same function. Also the high frequency circuit material used for this filter may not actually been used at a relatively high frequency. The material may have been used due to the tight dk values and the other properties mentioned.

Another item where most high frequency materials have advantage over traditional PCB circuit materials is minimal moisture absorption. Having low moisture absorption is important for several reasons and for high

frequency applications it is consistent dk. Water has a dk value of about 70 and even a small amount of moisture absorbed into a high frequency circuit or a controlled impedance circuit, can change the electrical performance.

Another application example will be a microstrip circuit, which will need to be formed to a specific shape around a mandrel that is ½" diameter. The electrical concerns are primarily low insertion losses (< 0.05 dB/in), tightly controlled impedance (50 ohms +/- 5%) and operating at 900 MHz. Typically an application operating at 900 MHz is not considered high frequency and more traditional PCB materials could normally be used. Yet considering the electrical concerns there are several reasons why a high frequency grade material should be used.

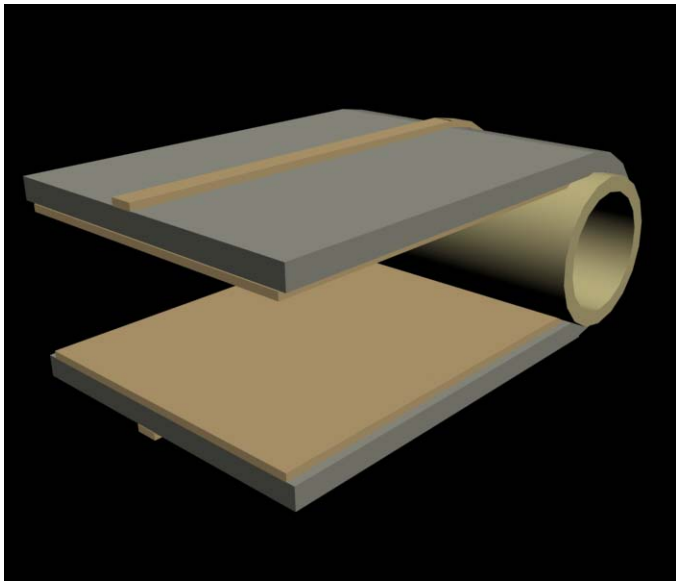


Figure 3. Second example of a simple microstrip, bent around a mandrel

The need for tightly controlled impedance translates to a substrate that must have tightly controlled thickness and dk values. Traditional PCB materials will not control these attributes near as well as high frequency circuit materials. Of course the etching control of the conductor width will need to be very well controlled at the circuit fabricator as well.

The concern of low insertion loss, overall, can be rather complicated. There are many issues that can affect this circuit property and the following are a few items that can be a concern: connectors, signal launch design, plating finish on the copper conductors, dissipation factor of the substrate, copper roughness, circuit geometry and some assembly processes. When choosing the material that will give the best advantage for insertion loss, then a material with low dissipation factor, smooth copper and low moisture absorption would be optimum. Also the circuit design should use a relatively thick substrate.

In this case there are concerns of a mechanical issue and forming a circuit around a mandrel. Typically a thinner substrate, with no glass reinforcement will be better for forming the circuit and generating less stress on the copper layers. Also the copper type should be rolled wrought or rolled annealed copper, which has a grain structure that is optimum for elongation in the x-y plane [1].

And one more consideration could be the type of plating finish to apply to the copper conductors of the microstrip. If enig (electroless nickel / immersion gold) were used, then the nickel is brittle and can be problematic for bending. Also the enig process will typically deposit a thickness of approximately 100 to 200 microinches of nickel and around 10 microinches of gold. With the operating frequency at 900 MHz, the skin effects will force the signal to use about 87 microinches of conductor. That means the signal energy will predominately use the nickel layer and nickel will cause an increase in conductor losses and ultimately insertion losses. This is due to the fact that nickel is less conductive than copper and that it has a permeability value that is much greater than copper. The permeability value will adversely affect the magnetic fields of the propagating waves.

In summary this application should use a high frequency material that has a tight control of the dk and thickness tolerance, low dissipation factor, smooth copper, non-glass reinforced, low moisture absorption and a non-nickel/gold finish. To determine the optimum thickness, there will need to be a trade-off between mechanical and electrical properties. For a one-time bend there is a rule of thumb that states the stress on the copper should be approximately 2% or less. Figure 4 shows a comparative table of different materials and thicknesses in regards to the mechanical stress values and insertion losses.

Model Order	Material	Thickness (in.)	Dielectric Constant, dk	Dissipation Factor, df	Conductor Width 50 ohms (in.)	Mechanical Stress (%)	Insertion Loss (dB/in)	Comment
1st	RO3003™	0.030	3.00 +/- 0.04	0.0013	0.075	7.583	0.0216	Stress too high
2nd	RO3003™	0.010	3.00 +/- 0.04	0.0013	0.025	2.335	0.0576	Stress is good, Insertion loss is bad
3rd	RT/duroid® 5880	0.010	2.20 +/- 0.02	0.0009	0.030	2.375	0.0335	Stress is good, Insertion loss is good

Figure 4. Comparison between different materials and thickness

In figure 4 it can be seen from the 1st model to the 2nd, the thickness of the substrate decreased. The decrease in thickness improved the mechanical stress significantly. A decrease in the substrate thickness will force a decrease of the conductor width in order to maintain 50 ohms. The decreased conductor width will increase the conductor loss and ultimately the insertion loss.

When going from the 2nd to the 3rd model a different material was selected which had a lower dielectric constant and lower dissipation factor, while maintaining the same substrate thickness. The lower dielectric constant would dictate an increase in conductor width to maintain 50 ohms. The increase in conductor width will lower conductor losses. And the decrease in dissipation factor will lower dielectric losses. With lower conductor and dielectric losses, then insertion loss will also decrease.

It can be seen that the stress number is a little higher than would be desired, but when looking at other models this appears to be the best-case scenario. In reality, this may be good for the actual application and if not then the diameter of the mandrel would need to increase some to lower the stress.

Some times there are interactions between the high frequency circuit materials and the PCB fabrication process, which can affect the circuit performance for the end user. Understanding the PCB fabrication concerns for these materials can be important for the end user. However it can be even more important for the circuit fabricator in order to achieve good manufacturing yields, a quality and highly reliable finished circuit. Each type of high frequency materials has their own unique circuit fabrication concerns.

The following will discuss the fabrication concerns for the most common high frequency materials; these materials are: PTFE (Teflon®), PTFE with ceramic fillers and non-PTFE thermoset resin systems with ceramic loading. Also the bonding materials that can be used to make multilayers from these materials will be discussed as well. Following will be information regarding LCP (Liquid Crystalline Polymer) materials, which are less commonly used and offer very unique properties to enable new evolving PCB applications. Lastly some hybrid circuit constructions will be discussed.

The high frequency circuit materials that have been used in the industry for the longest period of time are the PTFE materials. Most of these materials are not pure PTFE, but will have some small amount of micro-fiber glass impregnated into the substrate or will be PTFE with woven glass reinforcement or possible ceramic filled. In general and as a comparison to the other types of high frequency materials, the nearly pure PTFE without woven glass can be the most challenging type of circuit material to fabricate a PCB. There are several reasons for this and some of these are: PTFE has a high CTE, PTFE doesn't allow adherence of other materials easily, and the PTFE substrate is soft and can be easily distorted. From an electrical performance perspective the PTFE substrates are typically the best. The ceramic filled PTFE substrates are typically much easier for PCB fabrication.

The main issues for the PCB fabricator to know about for PTFE circuit fabrication are: drilling without any smear, never scrub or mechanically alter the substrate, dimensional stability (scaling) issues will have to be fine tuned and possibly adjusted on every panel, have very good practices to minimize handling damage of the soft

substrate, using a special through-hole wall preparation process to allow the copper plating to adhere to the PTFE drilled material and understanding how to laminate PTFE materials with other bonding materials.

There are no known processes that can de-smear PTFE, so when drilling the material it is of paramount concern to minimize heating and ensure no smearing of the substrate. General parameters for drilling PTFE substrates are shown in figure 5.

	Nearly Pure PTFE	Ceramic filled PTFE
Entry Material	Phenolic	Phenolic
Exit Material	Phenolic	Phenolic
Drill tool	Carbide	Carbide
Infeed	1-2mil/inch	2-3mil/inch
SFM	150-250	200-300
Retract rate	<500 inch/min.	<500 inch/min.
Drill life	~500-750 hits	~250-500 hits

Figure 5. Drilling parameters for PTFE substrates

The drill tool should be new and not a re-sharpened tool, for the nearly pure PTFE. This is to make sure that the cleanest possible cut in the material can be made without smearing. However a re-sharpened tool can be used for the ceramic filled PTFE substrates. If the circuit board is a hybrid using PTFE and other non-PTFE materials, then the drilling conditions should always be adjusted for the best PTFE drilling conditions. And if hybrid and only one outer layer of PTFE, then the circuit should be drilled with the PTFE up (toward drill tool entry).

After the holes have been drilled, the PTFE material will need to be prepared for activation so the copper plating process can achieve a good copper plated through hole. In the case of the nearly pure PTFE substrates, a wet chemistry process is recommended prior to the copper plating process. The process will use sodium naphthalene (or some derivative), which will strip a fluorine atom in order to make the PTFE substrate wetable and accept the copper plating. There are companies that supply this service and two of these are shown below with their contact information.

Poly-Etch®
Matheson Gas Products
61 Grove St
Gloucester, MA 01930
978/283-7700
Fax: 978/283-6177

™Fluoro-Etch®
Acton Associates, Inc
100 Thompson St
Pittston, PA 18640
570/654-0612
Fax: 570/654-2810

For the ceramic filled PTFE substrates, the same wet process treatment can be applied and there is a caution that a thorough bake must be done on the panels just prior to the copper plating. The ceramic filled PTFE substrates can absorb some of wet processing chemistry. If these chemistries are not baked out prior to sealing in the material with a PTH, then in later processes that are at elevated temperatures the entrapped chemistry will volatilize and cause delamination of the substrate. A safer process for the ceramic filled PTFE substrates would be the use of a special plasma cycle that can make the substrate wetable and accept plating chemistries. There are two plasma cycles that are commonly used. The parameters for the first plasma cycle are shown in figure 6.

Gasses	NH_3 or (70% H_2 / 30% N_2)
Pressure	100 mTorr Pump-down
Gasses	250 mTorr Operating
Power	4000 Watts
Frequency	40 KHz
Voltage	500 - 600 Volts
Cycle Time	10 - 30 minutes

Figure 6. Recommended plasma cycle for Ceramic filled PTFE substrates.

The second plasma cycle is similar to the first with all conditions except the type of gas that is used. In this case the gas would 100% Helium.

There are several materials that can be used as a bonding medium for multilayer PTFE circuits. Actually most bonding materials used in the PCB industry can be used as a bonding layer for a multilayer PTFE circuit, with some precaution. The main point of interest would be, not to alter the exposed substrate surface. After the copper etching process, the exposed PTFE substrate surface should not be mechanically altered in any way. The mirror image of the copper profile, from the copper that was etched away, will be the surface roughness of the exposed PTFE. This surface will need to remain unaltered in order to assist with some mechanical bonding with the bonding material. A scrubbing process will actually polish the soft PTFE surface and have an adverse affect on the bonding. And it is not necessary to perform a process on the exposed PTFE substrate to activate it in order to accept a bonding material. There is a benefit to performing a bake cycle for the ceramic filled PTFE, just prior to lamination. The bake cycle is intended to drive off any possibly absorbed processing chemistry and can be done at 121°C (250°F) for 1 to 2 hours. The recommended processing parameters for the particular bonding material should be followed.

The choice of bonding materials is a mixed decision between circuit fabrication issues and end use performance. If the bonding layer is a substrate layer that is not electrically important, then standard FR4 bonding materials can be used. If the layer is electrically important then a more high performance bonding material should be used. There are several bonding materials to choose from and in figure 7 there is information which highlights the electrical characteristics as well as some key fabrication issues for several high performance bonding materials.

Bonding Material	Dielectric Constant	Dissipation Factor	Lamination Temperature (F)	Preparation for PTH	Re-melt Temperature (F)
FEP	2.10	0.0010	565	Special	520
RO3003™	3.00	0.0013	700	Special	640
RO3006™	6.15	0.0020	700	Special	640
ULTRALAM® 3908	2.90	0.0025	554	Special	520
3001	2.30	0.0030	425	Special	350
RO3010™	10.80	0.0023	700	Special	640
RO4450B™	3.90	0.0040	350	Standard	N/A
RO4450F™	3.90	0.0040	350	Standard	N/A
SPEEDBOARD® C	2.60	0.0040	440	Special	640
FR4	4.50	0.0180	360	Standard	N/A

Figure 7. A list of high performance bonding materials

The mention of special preparation for PTH in figure 7 is in regards to making the substrate active to where it will accept copper plating, as previously discussed. The mention of a standard preparation is in regards to a standard permanganate or plasma cycle typically used for FR4 materials.

The reference of a re-melt temperature is for the thermoplastic materials that can reflow or melt in later processes at elevated temperatures. The re-melt could cause the multilayer to delaminate. If a soldering operation would

need to be performed on the multilayer it should be lower than the re-melt temperature or the proper bonding materials chosen to endure the soldering operation. The materials that have an "N/A" are thermoset and will not melt or reflow in subsequent processes.

Another family of high frequency materials would be the non-PTFE materials. These materials are basically a resin system with some filler and / or additive, which allows the materials to perform well at higher frequencies. Typically these materials are easier for PCB fabrication; however do not have the superior electrical performance as PTFE.

The materials that are a resin system with ceramic filler have unique demands for the fabricator. These demands do not require different equipment than what is needed to process FR4 materials, but different parameters for this equipment are often necessary.
The following is a brief highlight of the important fabrication issues regarding ceramic loaded resin systems: drill tool life is low, drill tools should not be re-sharpened or reused, rough drilled hole wall quality, loose ceramic particles after through hole wall preparation, prepreg's have minimal flow characteristics and the prepreg's are sensitive to low pressure areas during lamination.

Drilling the ceramic filled resin systems is more like excavating as opposed to drilling. At the through hole wall, the drill tool will either remove the ceramic particle it let it remain. This makes for a rough drilled hole-wall, which is actually better for plated copper adherence. Drilling the ceramic loaded materials will damage the drill tool quickly as well as the flute and this is why the drill tool should not be reused. The starting drill parameters for this type of material are shown in figure 8.

Surface Speed	300 - 500 SFM
Chip Load	0.002" - 0.004"/rev.
Retract Rate	500 IPM
Tool Type	Standard Carbide
Tool Life	2000 hits

Figure 8. Drilling parameters for non-PTFE ceramic filled resin systems.

Where minimizing heat to eliminate smear with PTFE is very important, that is not the case with ceramic filled resin systems. Of course it is best to have drilling conditions that yield a good quality through hole and with minimal smear. These materials can be processed in permanganate or a standard FR4 plasma cycle, for de-smear. The ceramic particles will not be affected by these processes, so it is necessary to have a high-pressure spray rinse after the de-smear process to remove any loose ceramic particles.

The prepreg's that are ceramic filled resin systems, will typically have less flow and are sensitive to low pressure areas during lamination. The low-pressure areas are many times due to the design of the circuit, where there are many copper features aligned in the cross-section of the circuit. This means that the areas between "stacked" copper features, will have lower pressure and this can cause the resin and ceramic particles to separate and not flow homogonously. The resin separation may have a different CTE than the homogonous prepreg and could have problems with solder or other elevated temperature exposures. To minimize this risk there are several items to consider: in the lamination process use a conformal material next to the panels, use highest pressure possible, at the beginning of the cycle have a hold for 20 minutes at the temperature where the prepreg will have the lowest viscosity and after which ramp up to the cure temperature. An example of a circuit with low-pressure areas and resin separation is shown in figure 9.

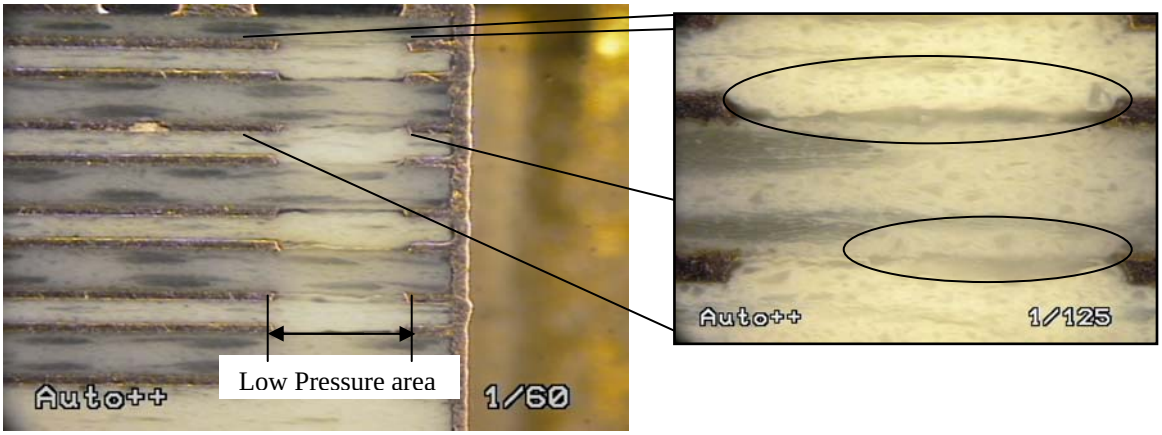


Figure9. Cross-section of a circuit with a low pressure areas and resin separation circled.

One particular ceramic filled resin prepreg, which was developed; to specifically minimize or eliminate the resin separation issue is the Rogers RO4450F™. Figure 10 shows a similar circuit using this prepreg.

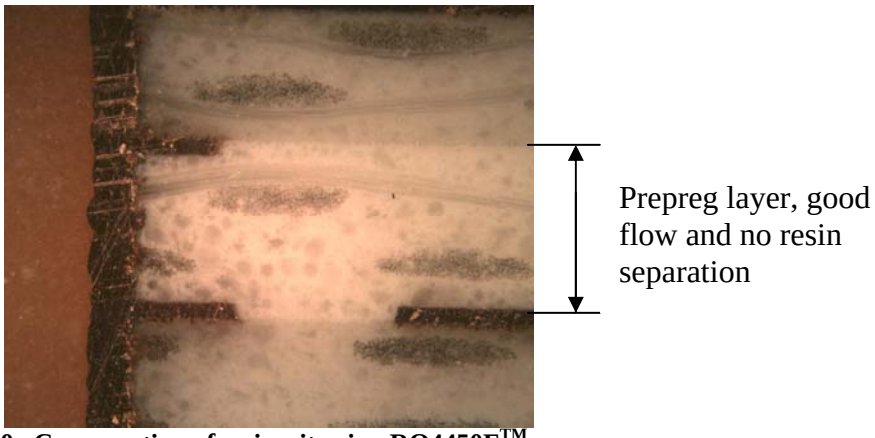


Figure10. Cross-section of a circuit using RO4450F™ prepreg

LCP circuit materials offer many unique characteristics for multiple end-user applications. These materials have been available in the industry for a number of years, however not well adopted by traditional PCB fabricators. The reason is due to unique processing requirements and some processes to be extremely well controlled.

Some of the excellent properties of the LCP material are: halogen-free, consistent dk, low TCdk, dk vs. frequency is very good, low dissipation factor, dissipation factor vs. frequency is very good, very high frequency capable, extremely low outgassing, extremely low moisture absorption, no issue with CAF or electromigration, very high MOT (Maximum Operating Temperature) rating, nearly perfect hermetic sealed circuit possible, excellent chemical resistance, inert substrate and is naturally flame retardant.
An example of a coplanar waveguide tested over a very wide frequency band and with different LCP thickness is shown in figure 11.

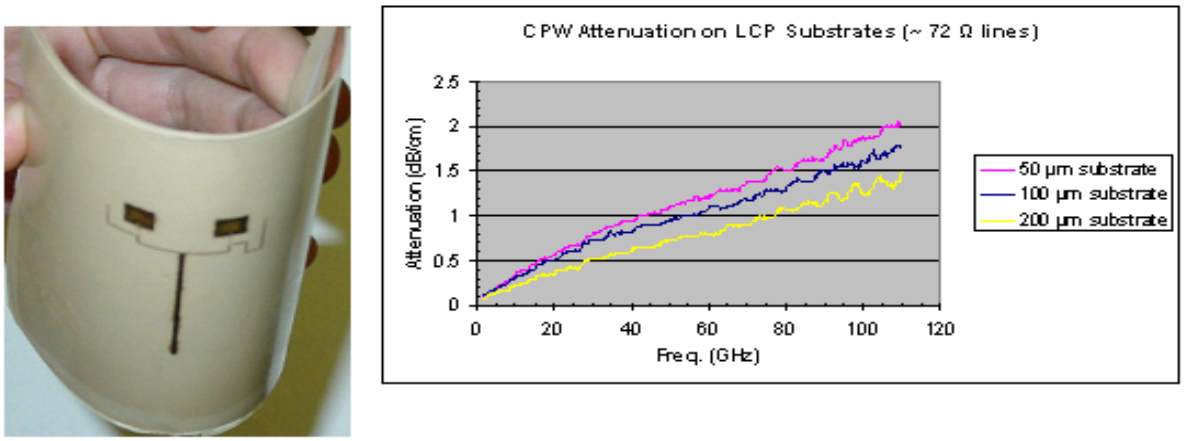


Figure 11. Flexible LCP circuit used as an antenna [2]

There are several fabrication issues and a more detailed discussion can be found at: http://www.rogerscorp.com/acm/products/17/ULTRALAM-3000-Series-Liquid-Crystalline-Polymer-Circuit-Materials.aspx

In general the main issues regarding LCP fabrication are: thin and soft laminates, dimensional stability (scaling) issues like thin flexible circuit materials, special high temperature lamination for LCP multilayers, special PTH preparation is necessary, drilling is important to avoid smear, venting and border patterns are important.

The lamination for a pure LCP multilayer will require a high temperature lamination that is well controlled for temperature and pressure distribution. The lamination materials that will be used next to the actual circuit materials will need to be very conformal. Typically several sheets of skived Teflon® (2 or 3 sheets of 2mil Teflon®) are used as the lamination release and the conformal. Besides the conformal having benefit of pushing the bond layer of the LCP into the circuit geometry, the conformal also helps to minimize any detrimental affect due to pressure distribution anomalies. There will be some small amount of outgassing during the lamination cycle and since LCP is a very good vapor barrier, having venting holes and good border channels are important. For the inner layer circuit border pattern, a dot pattern should be used and ensure that there are complete venting paths to the outside edge of the circuit panel. These dot patterns should not align from layer-to-layer. The venting holes should be drilled through all layers, as many as possible and are non-PTH holes. Prior to the high temperature lamination the LCP materials should have had a good acid rinse and a bake at 121°C (250°F) for 4 hours.

The high temperature lamination cycle uses a dwell at 260°C (500°F) with low pressure and vacuum assist in order to help remove the outgassing prior to raising the pressure and temperature for the fusion bond cycle. This is shown in figure 12.

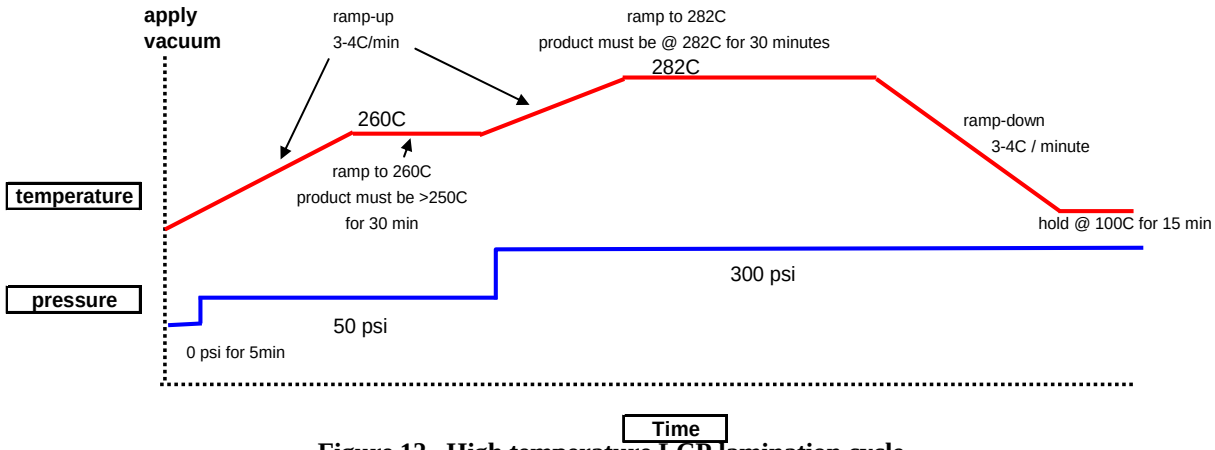


Figure 12. High temperature LCP lamination cycle

The drilling operation is also important and has similar concerns as drilling PTFE substrate. The main concern is to minimize the risk of smearing the substrate, which means to minimize the heat generation during the drilling process. Parameters for drilling LCP are shown in figure 13.

Chip Load	0.001" to 0.002"/rev.
Spindle Speed	200 to 500 SFM
Retract Rate	200 to 600 IPM
Entry Material	0.007" Aluminum
Exit Material	Phenolic

Figure 13. Drilling conditions for LCP multilayers

When drilling small holes or high aspect ratio then a peck drilling procedure may be needed. The maximum peck depth should not exceed 0.015". The drill tool should be high quality carbide and only use new tools.

The drilled hole-wall preparation for PTH can use either a chemical process or a plasma process. The chemical process uses a high concentration KOH. The plasma process is recommended and the parameters for this process are shown in figure 14.

	Gas Type, %						
Segment	CF_4	O_2	N_2	H_2	Vacuum, mTorr	Temperature, C	Time, min.
1	0	80	20	0	250	70	45
2	10	80	10	0	240	105	25
3	0	0	90	10	250	105	60

Figure 14. Plasma process for LCP substrates

There are many PCB applications where combinations of different circuit materials are used. Some applications will have high frequency circuit materials used on the PCB layers that are critical to electrical performance, while the other layers may use FR4 materials.

A common hybrid circuit will use one layer of a non-PTFE ceramic filled substrate for the layer 1 and 2, which will make up a high frequency microstrip transmission line. Then the other layers of the PCB will be more traditional PCB materials such as FR4 and are not electrically critical. With layer 1 being the signal and layer 2 the ground plane, then the bonding materials below layer 2 can be FR4 prepreg. These hybrids can typically be manufactured with good yields and assuming some caution is taken in a few processes. One area of concern would be the unbalance of material types and a possible warp issue. The most effective procedure to minimize the warp issue of the mixed materials is during the lamination cycle, specifically at the end of the cycle and after the prepreg is fully cured, then minimize the pressure to 50 psi and hold for 30 minutes. This low- pressure cycle is held while still at the cure temperature.

Another common hybrid circuit will use a combination of PTFE circuit materials and FR4. The critical electrical layers will use the PTFE substrate and the other layers would be the FR4. More often a ceramic filled PTFE substrate is used, because it will have a closer match to the FR4 thermal / mechanical properties and has a simpler PCB process. The PTFE substrate will typically not cause a warp issue, because the PTFE is very soft as compared to the FR4. During the lamination cycle the FR4 substrates will expand / shrink due to the temperature excursions and PTFE will be so soft that it will follow the FR4 movement. The drilling must be tailored to be optimum for the PTFE and the preparation for PTH will have several stages. The first stage is to de-smear and treat the FR4 material as necessary for the PTH preparation. The next stage will be to treat the PTFE for the PTH process. If a wet PTH preparation process is used for either the FR4 or the ceramic filled PTFE, then a bake at 121°C (250°F) for 1 to 2 hours is necessary just prior to the copper plating process.

A more exotic hybrid combination that has potential to be extremely good for high frequency applications and have mechanical flexibility is a special Rigid-Flex construction. This circuit would use the LCP materials for the flexible portion and would use the non-PTFE ceramic substrates for the rigid portion. This combination would have several advantages.

The transition from the rigid board areas to the flexible areas will not have connectors and the connection is built into the circuit. The lack of connectors means the cost for the connectors, assembly and reliability issues of connector goes away. Also if the design accounts for the transition from the rigid material to the flexible materials correctly there can be no impedance difference and that means a clean signal transition. For this type of circuit, the drill parameters will need to use the LCP parameters. The plated through hole preparation will be a several stage process as previously described. The last stage of the preparation process will be for the LCP materials.

In summary, the PTFE substrates are typically more difficult for circuit fabrication and will have superior electrical performance. The best material for electrical performance is the nearly pure PTFE substrates, such as Rogers RT/duroid® 5880. This material has been used for the most demanding electrical applications and it has very specific fabrication guidelines, which can be found at http://www.rogerscorp.com. Adding ceramic filler to the PTFE will lower the CTE, which is good for multilayer circuits and good PTH reliability. This addition also makes the material friendlier to PCB fabrication and an example of this material would be Rogers RO3003®. The non-PTFE materials are typically the most friendly to the PCB fabrication process and can have good electrical properties; an example of this material would be Rogers RO4350B™. The prepreg that is used with this material is RO4450F™ and that product has been optimized to minimize the risk of resin separation as previously shown in figure 10. Lastly the LCP materials offer a multitude of benefits to many applications, however the PCB fabrication process must be well controlled. An example of LCP material which is non-PTFE, non-filled, non-glass reinforced, halogen-free would be the ULTRALAM® 3850. That would be the laminate and the bonding material would be the ULTRALAM® 3908.

References:

[1] John Coonrod, “Bending Forming and Flexing Printed Circuits”, IPC / APEC Conference 2007.

[2] Thompson, Kirby, Papapolymerou, Tentzeris, “W-Band Characterization of Finite Ground Coplanar Transmission Line on Liquid Crystal (LCP) Substrates”, *IEEE Polytronic Conference 2003.*

Much of the fabrication information given can be found at: http://www.rogerscorp.com/acm/index.aspx

This paper is based on an article that appeared in the Printed Circuit Design and Fab, March and April 2009 issues

RT/duroid, RO4400, RO4450B, RO4450F, ULTRALAM and RO3003 are licensed trademark of Rogers Corporation
SPEEDBOARD® C is a trademark of W.L. Gore & Associates, Inc.©
Teflon® is a trademark of E.I. du Pont de Nemours and Company

High Frequency Circuit Materials used in the PCB Industry

John Coonrod, Rogers Corporation

High Freq materials

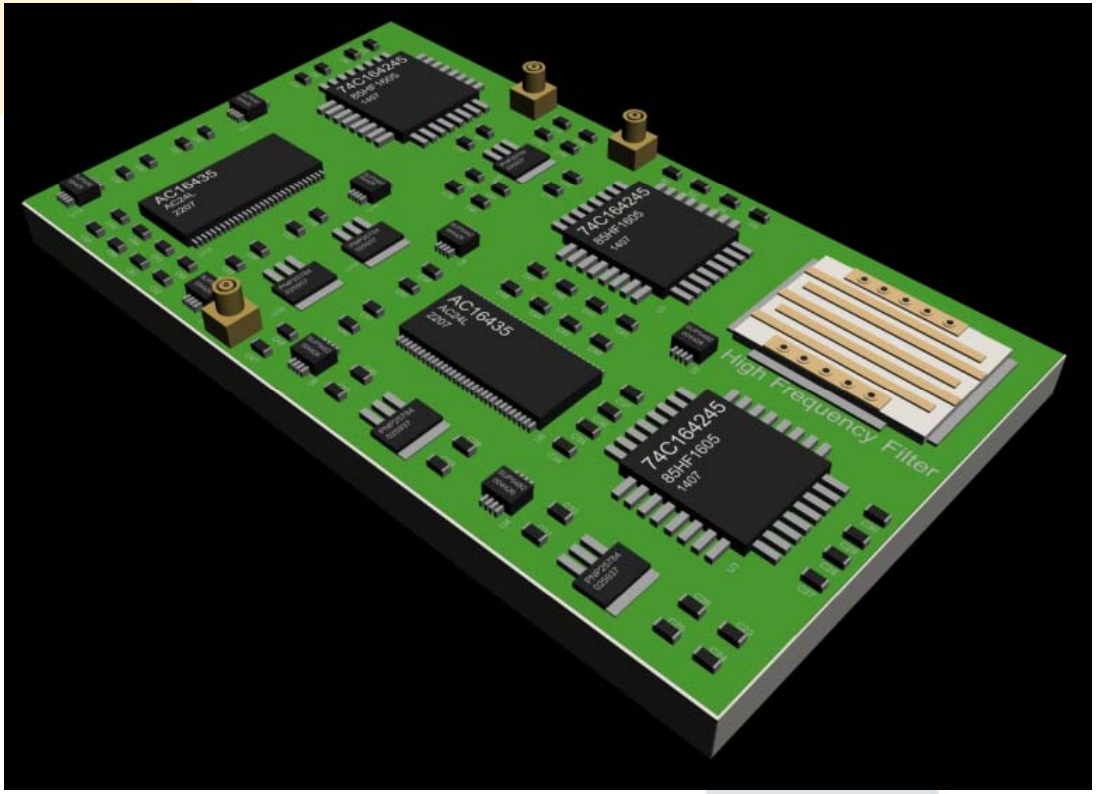
IPC MIDWEST CONFERENCE & EXHIBITION

Topics that will be covered:

- End-User, OEM Considerations
  - Understanding material properties as they relate to end-use applications
  - Critical properties
  - Other properties that may be critical and often disregarded
  - This material discussed by way of two application examples
- Fabrication considerations:
  - Guidelines for the following high frequency materials:
    - PTFE (Teflon®), PTFE with Ceramic filler, Non-PTFE Thermoset and LCP
  - Choosing the appropriate bonding materials for multilayers
  - Hybrid builds

- End-User, OEM Considerations
  - For a particular application, sometimes it is thought that only one material property is paramount
  - However many other material properties can be very important
  - Some of these are thought of initially in the design phase and some not
  - Some material properties can be altered by the environment

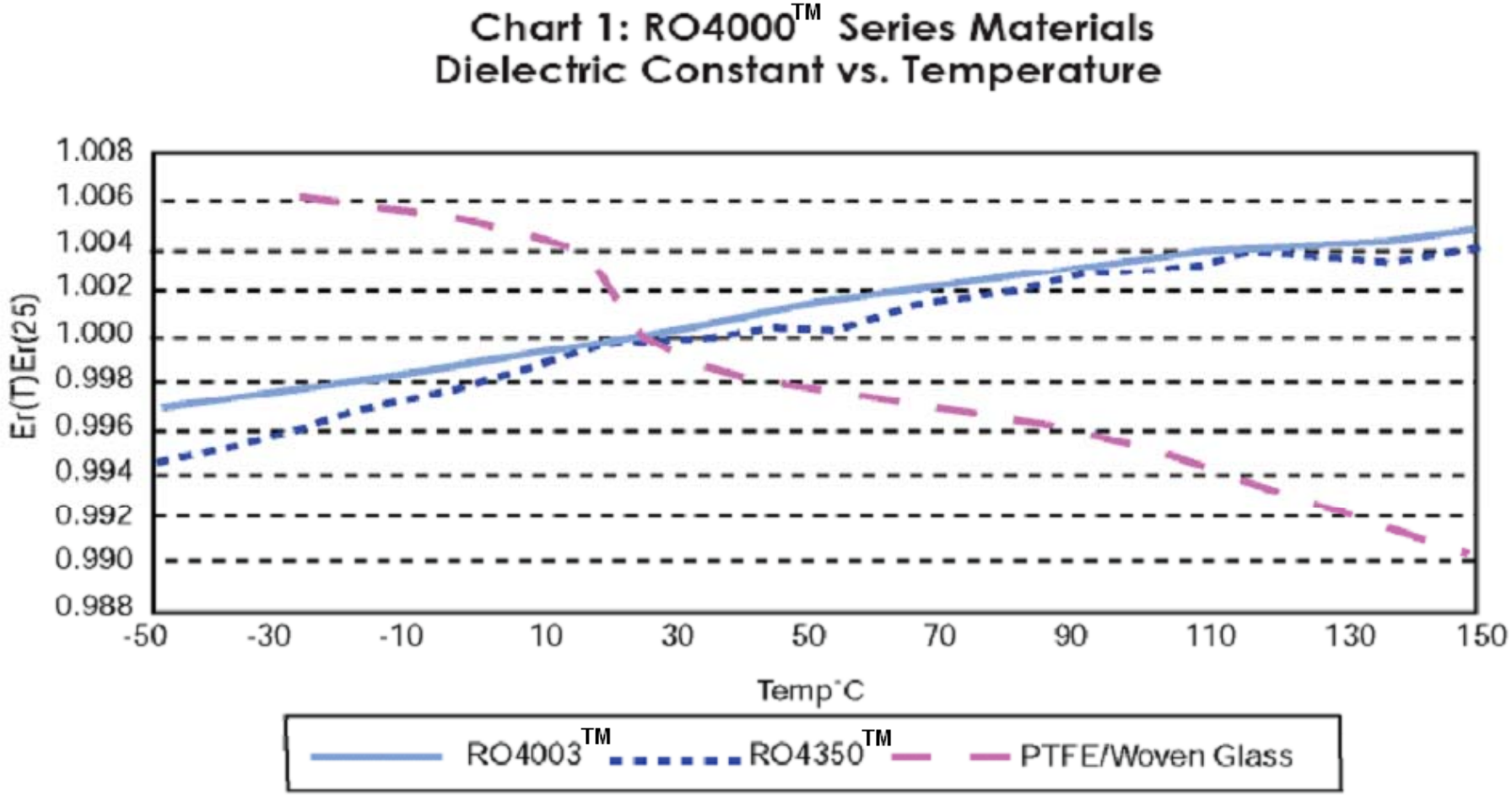
- End-User, OEM application example #1, Filter:
  - High frequency filter PCB, double-sided, PTH, microstrip edge coupled filter
  - Filter will be assembled to a FR4 PCB
  - The assembly will be sealed in a closure.
  - The enclosure will cycle from -25° C to +70° C



IPC MIDWEST CONFERENCE & EXHIBITION

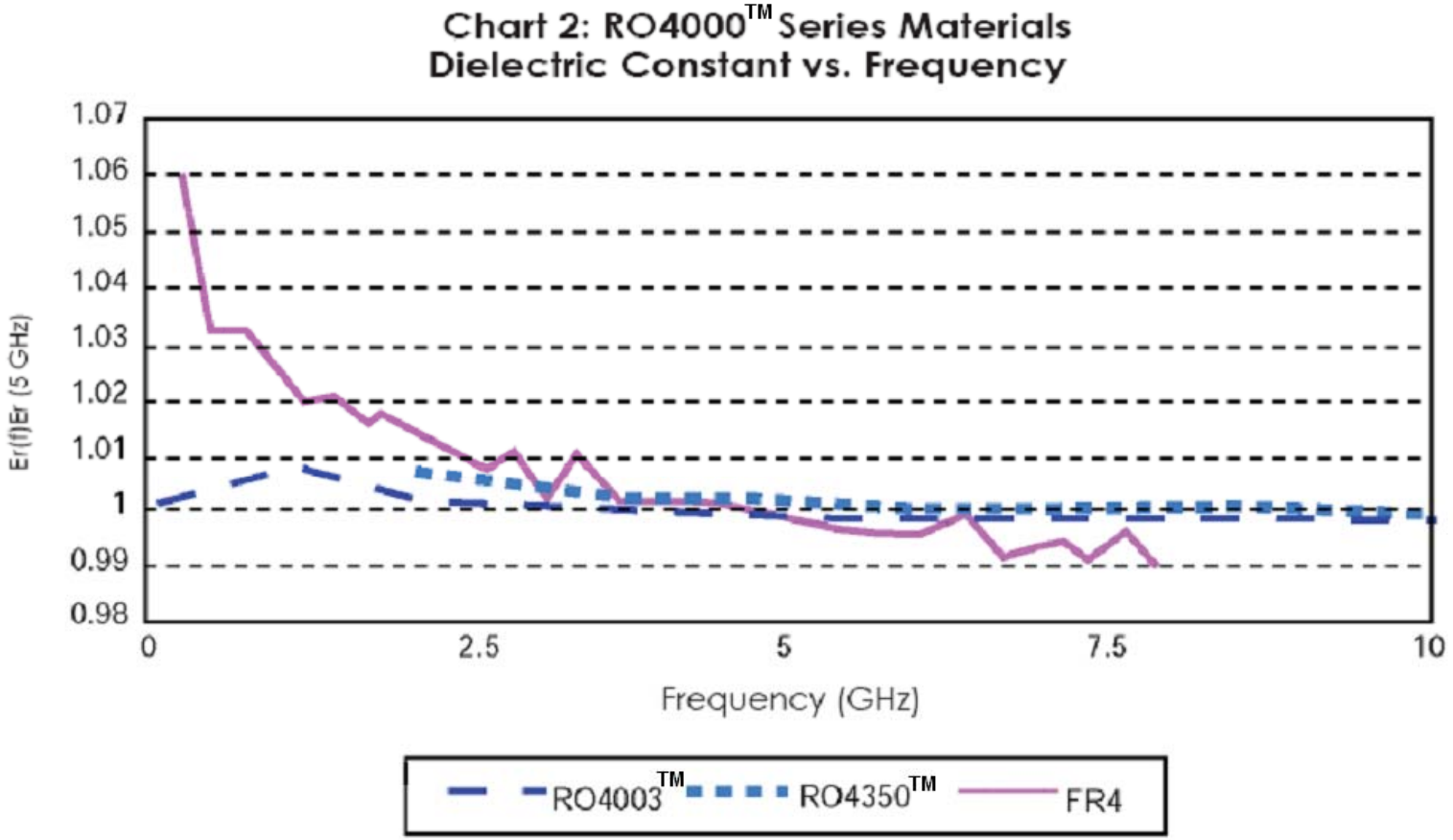
- End-User, OEM application example #1, Filter:
  - Edge coupled filters need very tight control of dk or ε_r (dielectric constant)
  - High Frequency laminate materials need to be consistent, lot-to-lot and within sheet
  - Other related material properties inter-related and possibly not considered:
    - TCdk, Thermal Coefficient of dielectric constant
      - This is how much the dk will vary with temperature
    - dk vs frequency attributes
      - All materials will have some variance of dk over a frequency sweep
    - Moisture absorption properties
      - Water has a dk value of approximately 70
      - If material can absorb much moisture, then dk can vary significantly

- End-User, OEM application example #1, Filter:
  - TCdk



IPC MIDWEST CONFERENCE & EXHIBITION

- End-User, OEM application example #1, Filter:
  - dk vs. Frequency



- End-User, OEM application example #1, Filter:
  - Other related material properties that may have impact on the performance and reliability for the end-use application
    - CTE (Coefficient of Thermal Expansion)
      - In this application it may "appear" that CTE of the filter circuit is not critical, but it could be a major reliability issue
        - If the x-y plane CTE of the filter is significantly different than that of the FR4 circuit a reliability issue may arise over time
        - Example: The assembly various repeatedly from 0° C to +30° C
          The CTE's are very different
          Major stress at the solder connecting the filter to PCB
          "Accordion effect" tears solder pads off PCB over time
    - Also the more well known z-axis CTE of the filter circuit can be a reliability issue regarding the PTH at assembly

- End-User, OEM application example #1, Filter:
  - In summary the high frequency laminate material should:
    - Have a very tightly controlled dk
    - A low TCdk
    - Stable dk vs. frequency curve, in the range of the intended application
    - Low moisture absorption
    - x-y plane CTE matched to the FR4 circuit material
    - Low z axis CTE

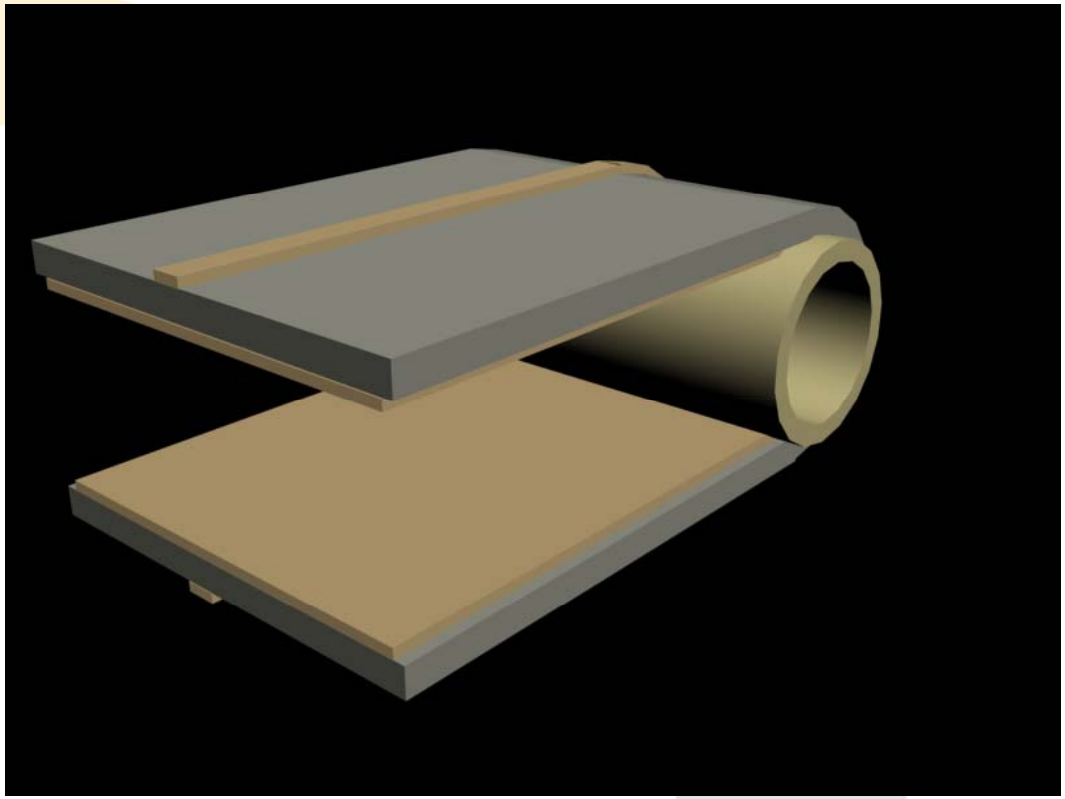
- End-User, OEM application example #1, Filter:
  - Some additional comments on the filter example:
    - If the physical size of the filter had to be minimized, then a high dk laminate could be used
      - All things equal, a higher dk will allow the circuit to shrink in size and still perform the same microwave function

Frequency 20 GHz, Filter using 1/4 wavelength segments

dk	Conductor length (in.)
2.20	1.100
3.66	0.887
6.15	0.697
11.00	0.528

    - And high frequency circuit materials are often used when the application is not high frequency due to:
      - tight control of dk for better yields of controlled impedance circuits
      - tight control of thickness for better controlled impedance circuits
      - Moisture absorption is very low, better yields in fabrication / assembly

- End-User, OEM application example #2, Long Microstrip transmission line:
  - Microstrip transmission line and the demands are:
    - Insertion loss must be less than 0.05 dB/inch
    - Circuit is bent around a 1/2" diameter mandrel
    - Control impedance to 50Ω ± 5%
    - Used at 900 MHz



IPC MIDWEST CONFERENCE & EXHIBITION

- End-User, OEM application example #2, Microstrip transmission line:
  - Typically 900 MHz is not considered high frequency
  - Due to other constraints, a high frequency material should be used because
    - Very low insertion loss is needed
    - Very tight control for impedance
  - Insertion loss can be quite complicated and several things have an effect
    - Connectors, circuit design of the signal launch, plated finish, dissipation factor of the substrate, copper roughness, circuit geometry and some assembly processes
  - Choosing a material for low insertion loss, it should have:
    - Low dissipation factor
    - Smooth copper
    - Low moisture absorption
    - Relatively thick substrate

- End-User, OEM application example #2, Microstrip transmission line:
  - Material considerations for the bending of a circuit
    - The material should be relatively thin
    - The copper should be smooth, rolled wrought or rolled annealed
    - Plated finish should not have brittle metal
    - No glass woven materials
    - Strain on the critical copper layer should be < ~2.0%
  - Plating finish concerns
    - Electroless nickel / immersion gold (enig)
      - Nickel is brittle and bad for bending
      - Nickel is much less conductive than copper; higher insertion loss
      - Nickel has a high value of permeability; higher insertion loss

- End-User, OEM application example #2, Microstrip transmission line:
  - To determine the optimum laminate thickness there is a trade-off between electrical and mechanical performance; three models ran:

Model Order	Material	Thickness (in.)	Dielectric Constant (dk)	Dissipation Factor (df)	Conductor width 50 ohm (in.)	Mechanical Strain (%)	Insertion Loss (dB/in.)
1st	RO3003™	0.030	3.00 +/- 0.04	0.0013	0.075	7.583	0.0216
2nd	RO3003™	0.010	3.00 +/- 0.04	0.0013	0.025	2.335	0.0576
3rd	RT/duroid® 5880	0.010	2.20 +/- 0.02	0.0009	0.030	2.375	0.0335

    - 1st model had good insertion loss, however the stress was too high
    - 2nd model, same material but thinner, had relatively good stress however the insertion loss was too high
      - A thinner substrate needs a narrower conductor to maintain 50Ω
      - Narrower conductor increases conductor loss and insertion losses
    - 3rd model, different material and same thickness, insertion loss is good and stress is relatively good
      - Lower dk means the conductor width increases, less conductor loss
      - Lower dissipation factor gives lower dielectric loss

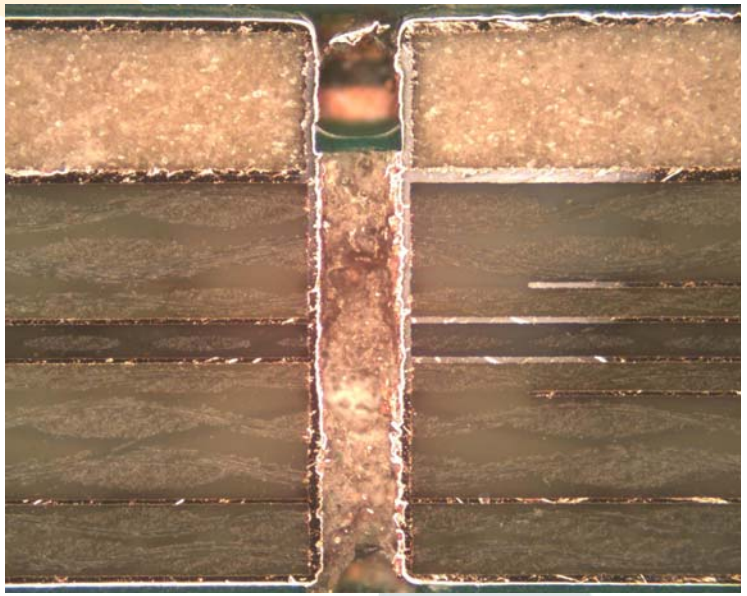
Electrical models done by Rogers Corporation MWI-2008 software. Mechanical modeling per mechanical beam composite theory [1]

- End-User, OEM application example #2, Microstrip transmission line:
  - Summary:
    - Sometimes high frequency materials are used at lower frequencies due to:
      - The tight control of dk and thickness of the material
      - Better for tight impedance specifications
      - Very low insertion loss is sometimes needed on long circuitry
        - Materials should have low dissipation factor
        - Smooth copper
        - Low moisture absorption
      - A thicker substrate is typically better for low insertion loss, however it can be problematic for bending a circuit
      - Plating finishes can have a detrimental effect on the mechanical and electrical behavior of a circuit

- Fabrication considerations:
  - There can be many interactions between high frequency PCB materials and the circuit fabrication and assembly processes
  - These interactions can affect the end-use properties
  - Understanding these possible interactions needs to be well understood for:
    - End-use performance
    - Reliability
    - Fabrication yields

IPC MIDWEST CONFERENCE & EXHIBITION

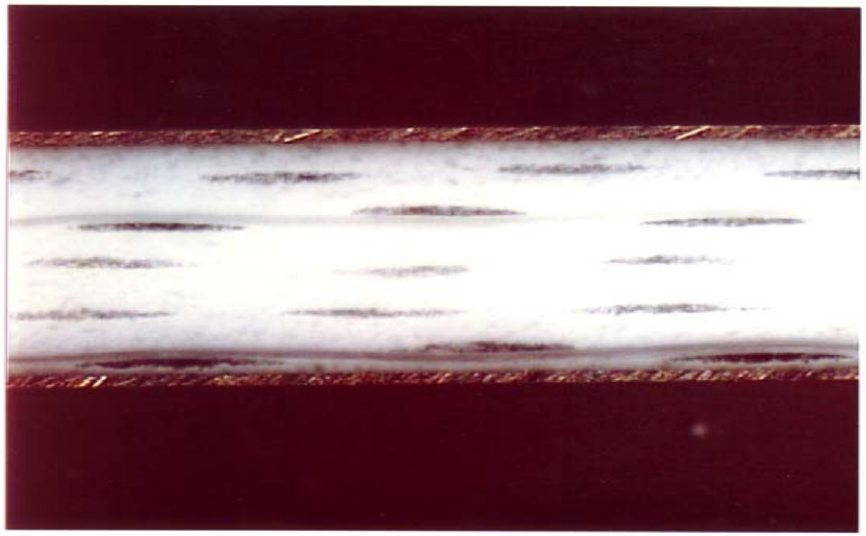
- Fabrication considerations:
  - Circuit Materials used in High Frequency Industry
    - PTFE (Teflon®) is the oldest high frequency materials used
    - PTFE with micro-fiber glass
    - PTFE with woven glass reinforcement
    - PTFE with ceramic filler
    - Non-PTFE thermoset resin systems with additives of fillers
    - LCP (Liquid Crystalline Polymer)



Hybrid, FR4 and top layer is ceramic loaded PTFE

- Fabrication considerations, PTFE:
  - Nearly pure PTFE substrates:
    - Difficult for circuit fabrication
    - Extremely good electrical performance
  - When ceramic filler is added to PTFE:
    - Circuit fabrication issues are eased
    - End-use electrical performance is slightly degraded
  - When ceramic filler and woven glass reinforcement is added to PTFE:
    - Circuit fabrication issues are eased much more
    - End-use electrical performance is more degraded, however it still can be quite good

- Fabrication considerations, PTFE:
  - Nearly pure PTFE substrates are difficult for circuit fabrication, due to:
    - PTFE has a high CTE
    - PTFE doesn't allow adherence of additive materials easily
    - PTFE is soft and can be easily distorted
  - The main issues regarding circuit fabrication of PTFE substrates are:
    - There are no known processes that can desmear PTFE
    - Drilling is critical to minimize heating to avoid smear
    - Never mechanically alter, stress or scrub the substrate
    - Dimensional stability (scaling) needs to be fine tuned
    - Handling practices are critical for the soft material
    - Special PTH preparation process is necessary
    - Multilayer builds can be a challenge due to CTE and choice of bonding matls



Non-PTFE high frequency material

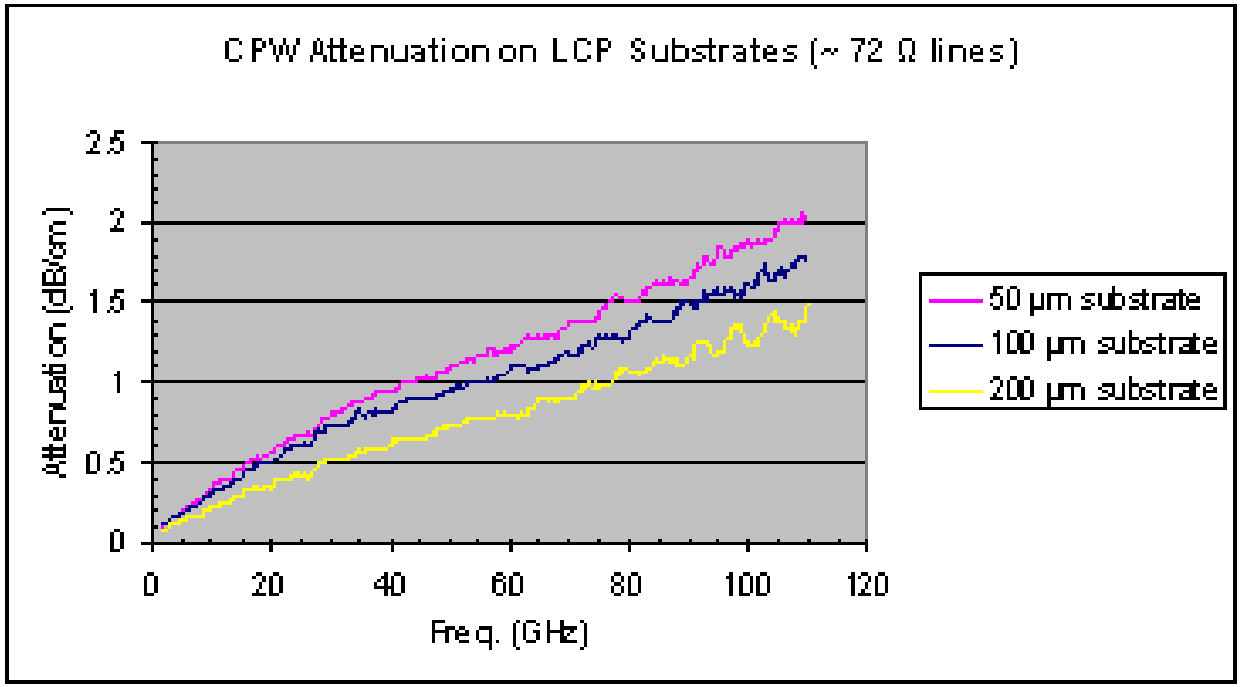
- Fabrication considerations, non-PTFE:
  - These materials are thermoset resin systems with an additive and /or fillers to achieve relatively good electrical performance
  - These materials are more user friendly to the PCB fabrication process
  - These circuit materials have good electrical properties, however the PTFE substrates typically have significantly better electrical performance

IPC MIDWEST CONFERENCE & EXHIBITION

- Fabrication considerations, non-PTFE:
  - In general these materials can use the same equipment as what is used for FR4
  - However different processing parameters than FR4 is necessary
  - Summary of key fabrication issues for this type of material:
    - Drill life is relatively low
    - Drill tools should be new and should not be re-used later
    - Rough drilled hole wall should be expected
    - Loose ceramic particles after through-hole preparation
    - Prepreg's have minimal flow characteristics
    - Prepreg's are sensitive to low pressure areas during lamination

- LCP (Liquid Crystalline Polymer) materials:
  - LCP materials have many unique circuit material properties
  - Have been available for many years, but not yet well adopted in the PCB industry
  - Very unique processing requirements
  - LCP attributes:
    - Halogen Free
    - Consistent dk vs. frequency
    - Low TCdk
    - Low dissipation factor
    - Very high frequency capabilities
    - Extremely low outgassing
    - No CAF or electromigration
    - Extremely low moisture absorption
    - Very high RTI or MOT
    - Nearly perfect hermetic material
    - Excellent Chemical resistance
    - Inert substrate
    - Naturally flame retardant

- LCP (Liquid Crystalline Polymer) materials:
  - Very high frequency capabilities

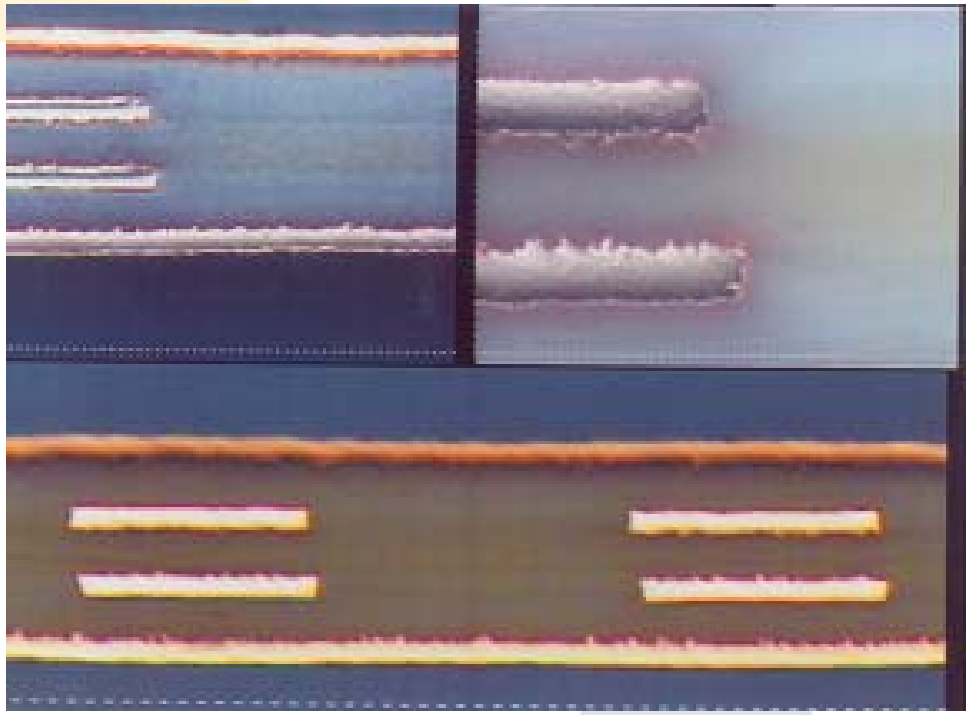


Flexible LCP circuit used as an antenna

From: [2] "W-Band Characterization of Finite Ground Coplanar Transmission Line on Liquid Crystal (LCP) Substrates"

IPC MIDWEST CONFERENCE & EXHIBITION

- LCP (Liquid Crystalline Polymer) materials:
  - Summary of the key fabrication issues regarding LCP:
    - Thin and soft laminates
    - Dimensional stability (scaling) issues, like thin flexible circuit materials
    - Special high temperature lamination for LCP multilayers
    - Venting and border patterns are important at lamination
    - Special PTH preparation is necessary
    - It is critical to avoid smear during drilling
    - Cleanliness prior to multilayer lamination is critical



IPC MIDWEST CONFERENCE & EXHIBITION

- Summary
  - Near pure PTFE substrates
    - Have the best electrical performance
    - Are less friendly to the PCB fabrication process
  - PTFE Ceramic filled with / without glass reinforcement
    - Have very good electrical performance, but not quite as good as pure PTFE
    - Are more friendly to the PCB fabrication process
  - Non-PTFE ceramic filled resin systems, High Frequency substrates
    - Have good electrical performance
    - Are very friendly to the traditional PCB fabrication process
    - Very high yields for the fabricator are possible

- References:
  - PTFE, Non-PTFE and LCP high frequency process guidelines:
    - http://www.rogerscorp.com/acm/index.aspx

[1] John Coonrod, "Bending and Forming High Frequency Printed Circuits", IPC Printed Circuits Expo®, APEX® and the Designers Summit, 2007.

[2] Thompson, Kirby, Papapolymerou, Tentzerix, "W-Band Characterization of Finite Ground Coplanar Transmission Line on Liquid Crystal (LCP) Substrates", IEEE Polytronic Conference 2003.

RT/duroid, RO4400, RO4450B, RO4450F, ULTRALAM and RO3003 are licensed trademarks of Rogers Corporation

The world runs better with Rogers. and the Rogers' logo are licensed trademarks of Rogers Corporation

SPEEDBOARD ® is a trademark of W.L. Gore & Associates, Inc.©

Teflon® is a registered trademark of E.I. Du Pont De Nemours and Company.

This presentation is based on an article that appeared in the Printed Circuit Design and Fab, March and April 2009 issues

The information in this presentation is intended to assist you in working with Rogers' High-Frequency Materials. It is not intended to and does not create any warranties, express or implied, including any warranty of merchantability or fitness for a particular purpose or that any results show in this presentation will be achieved by a user for a particular purpose. The user is responsible for determining the suitability of Rogers' High Frequency Materials for each application.

This presentation may not be reproduced, copied, published, broadcast, transmitted or otherwise distributed without the written approval of Rogers Corporation.

©2009 Rogers Corporation. All rights reserved

IPC MIDWEST CONFERENCE & EXHIBITION

- Fabrication considerations. PTFE:
  - Drilling:
    - Minimize heat during drilling
    - Some trial & error testing may be beneficial to determine different drill stack-ups
    - With nearly pure PTFE, new tools only to be used (no resharpened tools)
    - Ceramic filled PTFE can use resharpened tools
    - If a hybrid build and only one layer of PTFE substrate on the outside layer, then drill with the PTFE substrate up (toward drill entry)
    - If a hybrid build then the drilling conditions should be for the PTFE
    - Starting point drilling conditions:

	Nearly Pure PTFE	Ceramic filled PTFE
Entry Material	Phenolic	Phenolic
Exit Material	Phenolic	Phenolic
Drill tool	Carbide	Carbide
Infeed	1-2mil/inch	2-3mil/inch
SFM	150-250	200-300
Retract rate	<500 inch/min.	<500 inch/min.
Drill life	~500-750 hits	~250-500 hits

IPC MIDWEST CONFERENCE & EXHIBITION

- Fabrication considerations, PTFE:
  - PTH preparation:
    - Nearly pure PTFE
      - It is highly recommended to use a wet process to strip a fluorine atom such as sodium naphthalene or a derivative
      - Companies that supply this service are:

Poly-Etch®
Matheson Gas Products
61 Grove St
Gloucester, MA 01930
978/283-7700
Fax: 978/283-6177

™Fluoro-Etch®
Acton Associates, Inc
100 Thompson St
Pittston, PA 18640
570/654-0612
Fax: 570/654-2810

      - If a hybrid build, then treat the other circuit materials accordingly and leave the sodium treatment for last, just prior to copper plating

- Fabrication considerations, PTFE:
  - PTH preparation:
    - Ceramic filled PTFE
      - The same sodium treatment can be used with a caution
      - Ceramic filled PTFE has some porosity and can absorb processing chemistries
      - A bake after any wet process is a good safe practice
      - After the sodium treatment and just prior to copper plating, bake at 250° F for 1 hour
      - A plasma cycle is safer for these types of substrates
      - Two cycles are recommended
      - The cycle shown to the right:
      - Same cycle except using 100% Helium
      - Hybrid; PTH preparations for other materials 1st

Gasses	NH_3 or (70% H_2 / 30% N_2)
Pressure	100 mTorr Pump-down
Gasses	250 mTorr Operating
Power	4000 Watts
Frequency	40 KHz
Voltage	500 - 600 Volts
Cycle Time	10 - 30 minutes

IPC MIDWEST CONFERENCE & EXHIBITION

- Fabrication considerations, PTFE:
  - Multilayer bonding
    - Most bonding materials used in the PCB industry can be used
    - It is critical to not alter the exposed PTFE substrate surface prior to bonding
      - The mirror image of the copper will be the substrate surface profile and aids to mechanical boding
      - A scrub will actually polish the soft substrate surface and can be problematic
      - It is not necessary to sodium treat or plasma the exposed PTFE surface prior to bonding
      - A bake at 250° F for 1 hour for the ceramic filled PTFE is suggested

- Fabrication considerations, PTFE:
  - Mulitlayer bonding materials choice
    - Mixed decision between fabrication issues and end-use performance
    - If the bonded layer is non-electrically critical then standard FR4 prepreg can be used
    - If the bonded layer is electrically critical then a high performance bonding material is necessary and several follow:

Bonding Material	Dielectric Constant	Dissipation Factor	Lamination Temperature (F)	Preparation for PTH	Re-melt Temperature (F)
FEP	2.10	0.0010	565	Special	520
RO3003™	3.00	0.0013	700	Special	640
RO3006™	6.15	0.0020	700	Special	640
ULTRALAM® 3908	2.90	0.0025	554	Special	520
3001	2.30	0.0030	425	Special	350
RO3010™	10.80	0.0023	700	Special	640
RO4450B™	3.90	0.0040	350	Standard	N/A
RO4450F™	3.90	0.0040	350	Standard	N/A
SPEEDBOARD® C	2.60	0.0040	440	Special	640
FR4	4.50	0.0180	360	Standard	N/A

IPC MIDWEST CONFERENCE & EXHIBITIO

- Fabrication considerations, PTFE:

Bonding Material	Dielectric Constant	Dissipation Factor	Lamination Temperature (F)	Preparation for PTH	Re-melt Temperature (F)
FEP	2.10	0.0010	565	Special	520
RO3003™	3.00	0.0013	700	Special	640
RO3006™	6.15	0.0020	700	Special	640
ULTRALAM® 3908	2.90	0.0025	554	Special	520
3001	2.30	0.0030	425	Special	350
RO3010™	10.80	0.0023	700	Special	640
RO4450B™	3.90	0.0040	350	Standard	N/A
RO4450F™	3.90	0.0040	350	Standard	N/A
SPEEDBOARD® C	2.60	0.0040	440	Special	640
FR4	4.50	0.0180	360	Standard	N/A

the PCB Industry

- "Standard" preparation refers to a standard FR4 process such as permanganate or CF_4 plasma cycle
- Re-melt temperature refers to the temperature where the thermoplastic material will start to reflow
  - A multilayer that has to be exposed to elevated temperatures later (assembly) should consider the re-melt temperature to avoid delamination

IPC MIDWEST CONFERENCE & EXHIBITION

- Fabrication considerations, non-PTFE:
  - Drilling:
    - Drilling the ceramic filled substrate will damage the tool relatively quick
    - Drilling the ceramic filled substrate will damage the flute of the tool and that is why the tool should not be re-used
    - When drilling this material, it should be thought of as excavating as opposed to drilling
      - The drill tool will either remove the ceramic particle or allow it to remain as part of the hole wall
      - Rough hole wall can be better for copper plating adherence
    - When establishing the drilling process for this material, the suggested starting parameters follow:

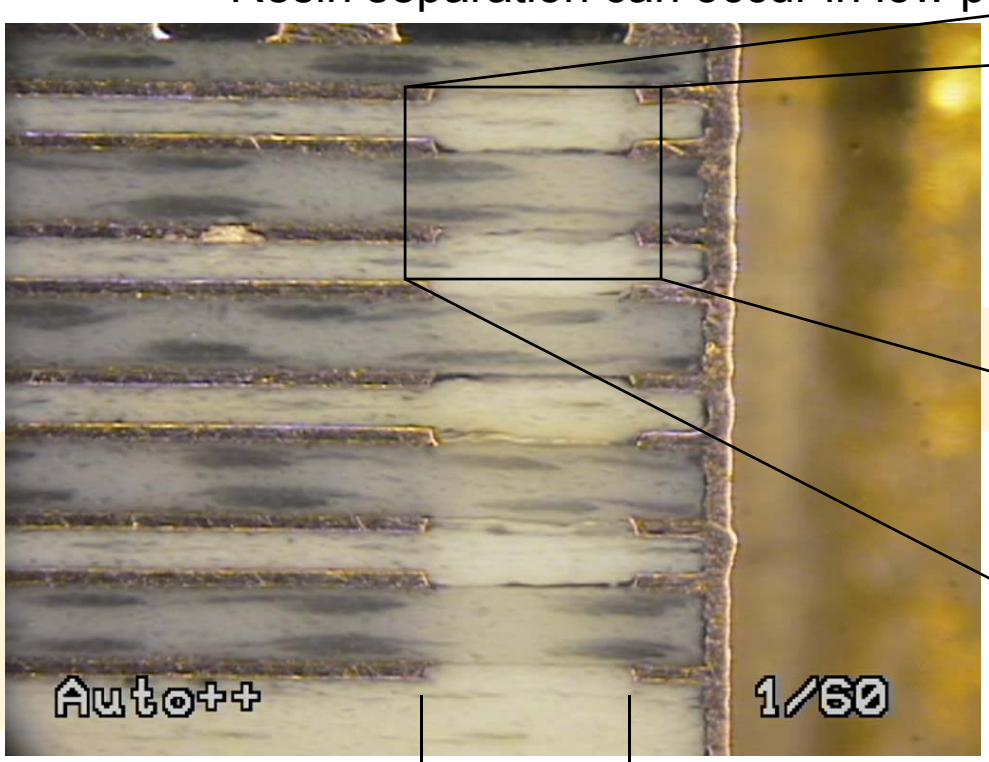
Surface Speed	300 - 500 SFM
Chip Load	0.002" - 0.004"/rev.
Retract Rate	500 IPM
Tool Type	Standard Carbide
Tool Life	2000 hits

IPC MIDWEST CONFERENCE & EXHIBITION

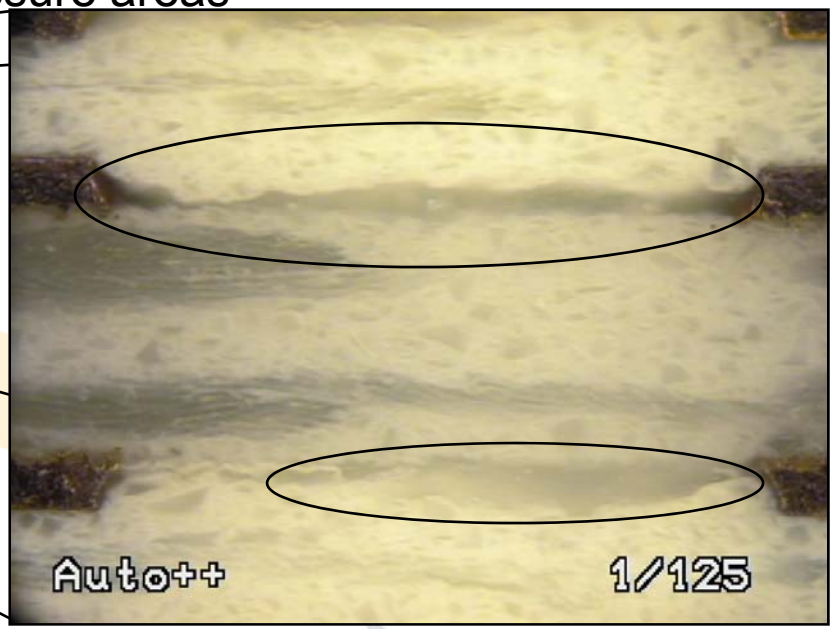
- Fabrication considerations, non-PTFE:
  - Desmear:
    - Permanganate can be used however about 25% more exposure is typically needed
    - A standard FR4 plasma cycle using CF_4, N_2, O_2 can be done
    - When using either of these methods to desmear or just clean the drilled hole, a high pressure spray rinse should follow
      - The ceramic filler is not affected by the desmear and it is possible to have loose ceramic particles on the hole-wall

- Fabrication considerations, non-PTFE:
  - Prepreg's and lamination:
    - Ceramic filled non-PTFE prepreg's typically have very low flow characteristics
    - Generally can not be used for via fill
    - Higher pressure is needed during lamination
    - Can be problematic for thicker inner layer copper or thick plated up sub's
    - Low pressure areas during lamination can be a significant issue

- Fabrication considerations, non-PTFE:
  - Prepreg's and lamination:
    - Low pressure areas due to the circuit construction and stack-up of copper
    - Resin separation can occur in low pressure areas



Low Pressure area

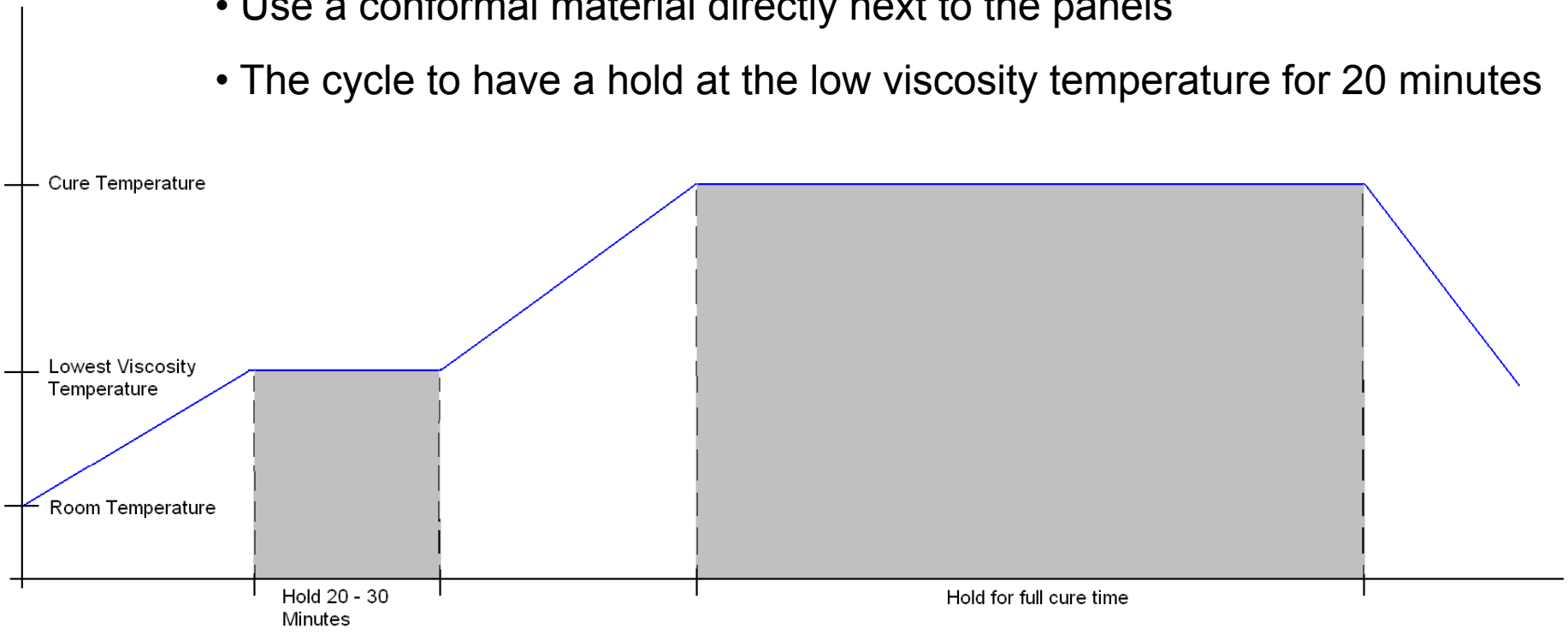


Circled resin separation areas

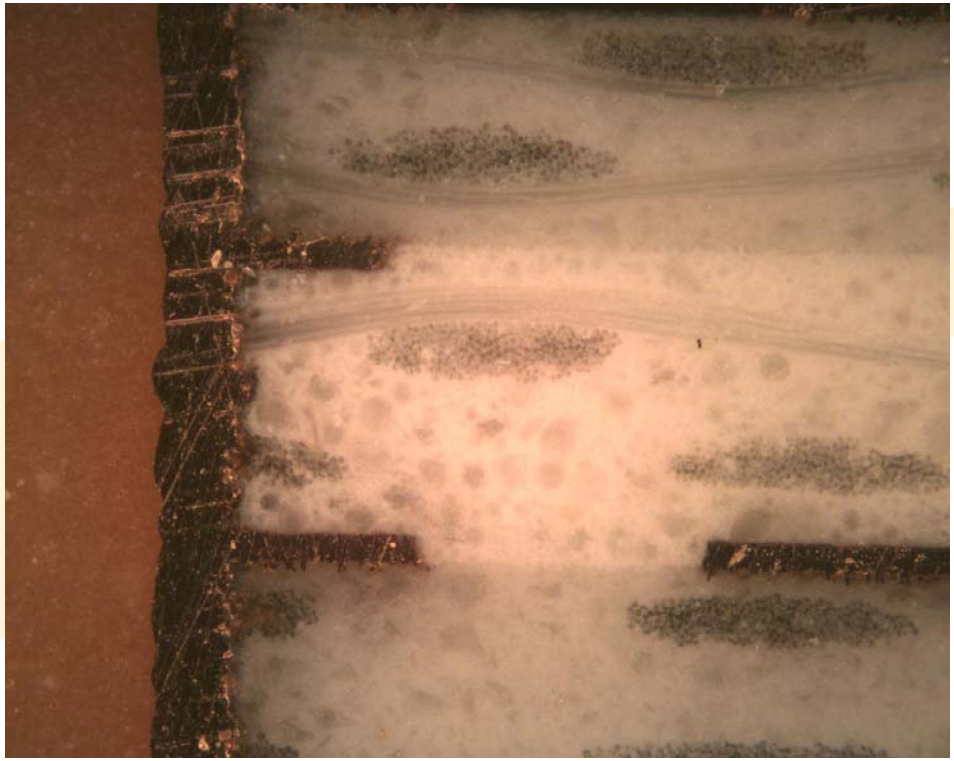
- The resin separation (non-homogeneous prepreg) will not have the correct CTE and later thermal exposure could be an issue

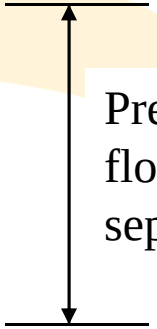
IPC MIDWEST CONFERENCE & EXHIBITION

- Fabrication considerations, non-PTFE:
  - Prepreg's and lamination:
    - To minimize the risk of resin separation:
      - Use highest pressure possible
      - Slow rate of rise
      - Use a conformal material directly next to the panels
      - The cycle to have a hold at the low viscosity temperature for 20 minutes



- Fabrication considerations, non-PTFE:
  - Prepreg's and lamination:
    - A special prepreg was developed and proven to greatly minimize or eliminate the resin separation issue
    - Rogers' RO4450F™ prepreg



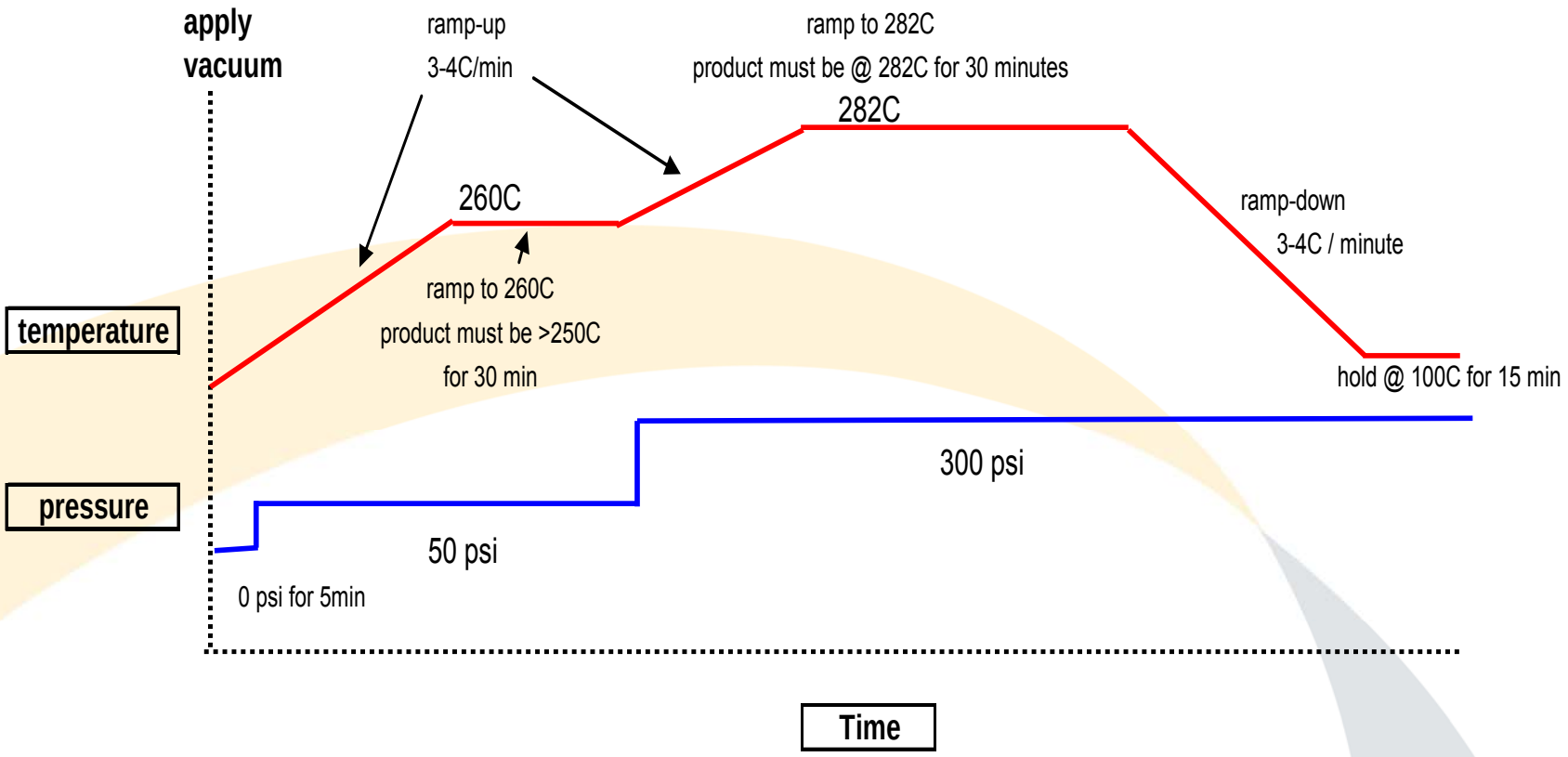


Prepreg layer, good flow and no resin separation

IPC MIDWEST CONFERENCE & EXHIBITION

- LCP (Liquid Crystalline Polymer) materials:
  - Special high temperature lamination for LCP multilayers
    - Just prior to lamination a thorough chemical (acid rinse) clean is necessary
    - Pre-bake before lamination at 250° F (121° C) for 4 hours
    - Venting and border patterns are important at lamination
      - Venting holes (non-PTH) should be used in all layers of the circuit
      - Border patterns to be a dot pattern that doesn't align in the z-axis
    - High conformal material used next to the panels
      - several layers of skived Teflon® lamination release is recommended
    - The lamination cycle should have a hold at 500° F (260° C) for 30 minutes at minimal pressure; then ramp up to the final lamination temperature

- LCP (Liquid Crystalline Polymer) materials:
  - Special high temperature lamination for LCP multilayers
    - Lamination cycle



IPC MIDWEST CONFERENCE & EXHIBITION

- LCP (Liquid Crystalline Polymer) materials:
  - Drilling
    - Similar concerns with drilling PTFE; do not smear while drilling
    - Drilling small holes or high aspect ratio may require peck drilling
      - Maximum peck depth should not exceed 0.015"
  - Plated through hole preparation
    - Wet process
      - Hot KOH is effective, however a possible safety / environmental issue
      - A special plasma process is recommended:

	Gas Type, %						
Segment	CF_4	O_2	N_2	H_2	Vacuum, mTorr	Temperature, C	Time, min.
1	0	80	20	0	250	70	45
2	10	80	10	0	240	105	25
3	0	0	90	10	250	105	60

- Hybrid Builds
  - Many applications have 2 or 3 conductive layers that are electrically critical and the rest of the circuit is not electrically critical
  - The critical layers can use the High Frequency specialty material, whereas the non-critical layers can use more traditional FR4 type of materials
  - Some hybrid circuits
    - PTFE substrates combined with FR4
    - Non-PTFE High Frequency substrates combined with FR4
    - High performance Rigid-flex hybrid

- Fabrication concerns for Hybrid Builds
  - PTFE + FR4 hybrid
    - Warp is typically not an issue
      - During elevated temperature process, the PTFE is so soft that it will move to where the other circuit materials pushes it from their CTE and/or stress
    - Drilling should be fine tuned and the key is to ensure that the PTFE does not smear
    - If the PTFE is only for layers 1 and 2, then drill with the PTFE toward the drill
    - Plated through hole preparation:
      - Perform the preparation for the other circuit materials first, such as permanganate for the FR4 layers. Then perform the PTFE preparation.
      - If it is ceramic filled PTFE it is highly recommended that the PTH preparation is plasma. If not, perform a bake just prior to copper plating
    - If the bonding layer is non-electrically critical, then use traditional PCB prepreg such as FR4. This would be true if layers 1 and 2 were high frequency microstrip and the other layers are bonded below layer 2 as a ground plane

- Fabrication concerns for Hybrid Builds
  - non-PTFE + FR4 hybrid
    - Warp can be an issue
      - Construction should be as balanced as possible
      - If only one layer of non-PTFE is critical, it should be in the middle of the circuit stack-up
      - If warp is an issue, then use a bumped cycle in lamination
        - While in the lamination press, after the full cure has been performed, reduce the pressure to minimum, hold at temperature for 30 minutes
    - Plated through hole preparation:
      - Perform the preparation for the other circuit materials first, such as permanganate for the FR4 layers. Then perform the non-PTFE prep
      - It is highly suggested to use the plasma process
    - If bonding layer is non-electrically critical, then use traditional PCB prepreg such as FR4. If layers 1 and 2 were high frequency microstrip with layer 2 as a ground plane then use standard FR4 prepreg

- Fabrication concerns for Hybrid Builds
  - High Performance Rigid-flex
  - Combination of flexible high frequency materials and high frequency rigid materials
  - The flexible portion could use:
    - Thin non-glass woven PTFE substrates
    - LCP
  - The rigid portion could use a variety of high frequency materials, however it may be better to use a material that has simpler requirements for circuit fabrication
  - A combination of LCP and RO4350™ laminate has been used successfully
  - Benefits
    - Cost, real estate, reliability issues and signal integrity are reduced by not having a connector
    - If the design is done correctly regarding the transition from the rigid to flex, no impedance mismatch and a clean signal transition