Electronic Design® LIBRARY

A compendium of articles from *Electronic Design*

HOW MATERIAL SELECTION CAN MAKE OR BREAK YOUR PCB

ElectronicDesign. LIBRARY

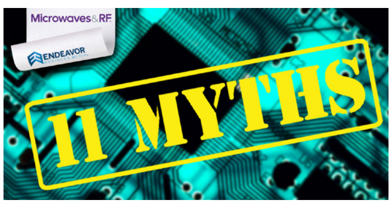
Sponsored by Waters™ | TA Instruments

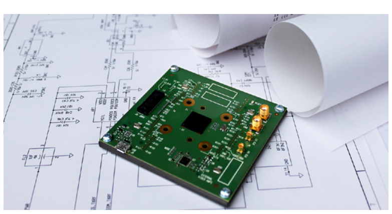
INTRODUCTION

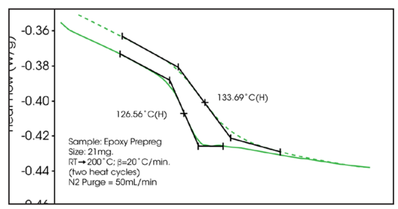
PRINTED CIRCUIT BOARDS (PCBS) dominate the use of electronics from tiny PCBs in smartwatches to massive PCBs used in cloud servers. But not all PCBs are created equal. Developers need to take into account a variety of considerations, including production standards, material impact on performance, and application conditions. This eBook takes a look at many of these issues and delves into details from epoxy curing to myths about the industry.

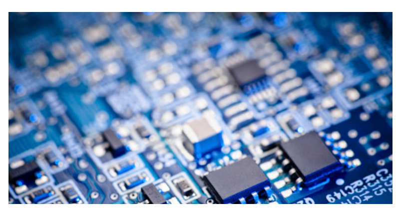
Bill Wong
Editor,
Senior Content Director, Electronic Design & MWRF

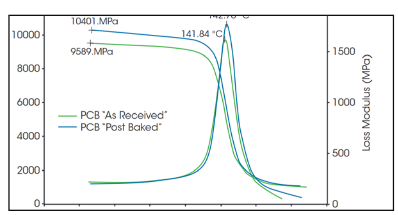
CONTENTS

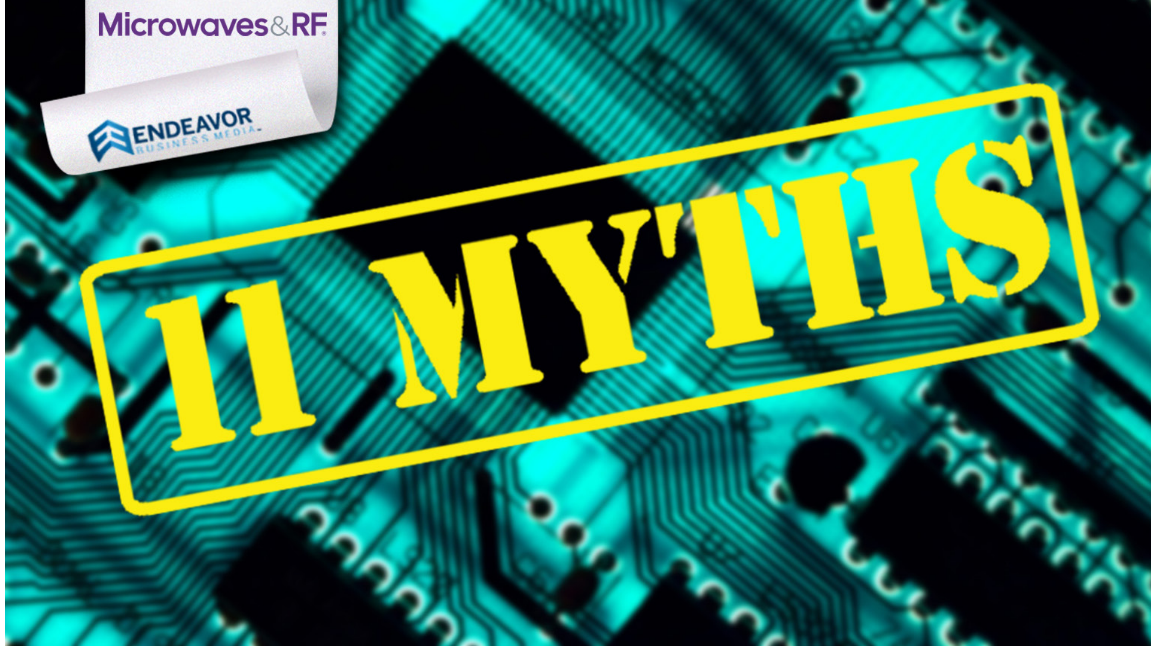












Dreamstime: Colicaranica

CHAPTER 1:

11 Myths About the PCB Industry

KELLA KNACK, Director of Strategic Marketing, *Avishtech*

Today's high-data-rate, high-frequency designs create new challenges for printed-circuit-board design.

Printed circuit boards (PCBs) have been around so long that product developers often think their design, manufacturability, and reliability is a "no-brainer." It's widely assumed that the value in any modern electronic product is driven by the components mounted on the board and the software applications embedded within these components.

However, today's high-frequency, high-data-rate devices are challenging these long-held assumptions such that a successful PCB implementation has once again become one of the focal points in the overall product development process. Below are some long-held myths associated with board development.

1. PCBs are commodities and essentially just the "carriers" for the ICs mounted onto them.

Yes, it's true. For a long time, PCBs were not a huge area of concern. There was lots of "wiggle room" in the roles that various aspects of board technology and materials played in the overall end-product design, so getting the PCB design "kind of right" often sufficed. But, as we began moving up the speed curve (starting at about 2.5 Gb/s), any margin for error in board design disappeared. As we climb further up the curve, such latitude for creating a board that works right the first time and every time thereafter shrinks.

Now with today's high-frequency, high-data-rate designs, the tolerance for any kind of misstep has essentially evaporated. More importantly, there are no "do-overs" with today's fine-pitch components. If your board fails, it's not just the board you lose. You're also losing components that can cost tens of thousands of dollars. On top of that, there's the loss of revenue resulting from being late to market, or not having a sufficient service life to provide long-term revenue and profitability.

2. The PCB design process is very straightforward.

Of course, certain electronic devices don't require a complex board design (anything more than eight layers). But for today's high-frequency, high-data-rate products, PCBs are the critical foundation upon which all other design development requirements are built. **Figure 1** depicts the flow of a typical design and fabrication process. As shown, PCB design and fabrication involve a multistep process wherein errors can occur in any stage.

Think of it in terms of building a house. The foundation may initially look fine, but if it's not successfully designed and built from the outset, building atop it could result in a catastrophic failure of the whole structure.

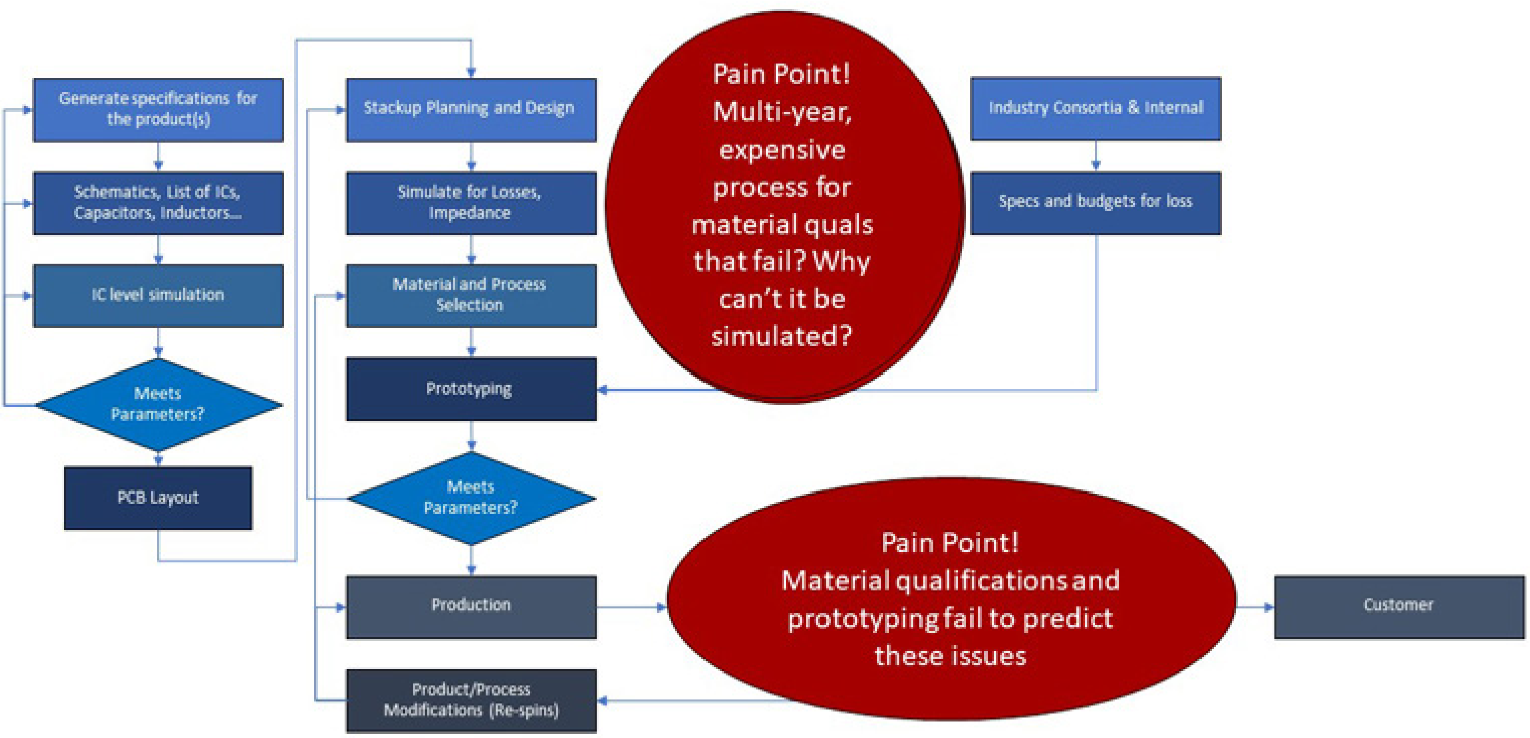
3. PCBs don't contribute to the overall operation of the end products into which they're incorporated.

In complex designs, the PCB's design and the characteristics of the constituent laminate material play a huge role in the end product's performance. The three main laminate characteristics that factor into design, manufacturability, reliability, and lifespan of a product are impedances, losses, and skew.

However, each of those characteristics has subsets, and in the final analysis at least 10 different material characteristics come into play when you're developing a high-frequency, high data-rate design. Ignore them at your own peril—any one of them might be enough to throw your entire product design off the rails.

4. EDA tools make PCB design a "pushbutton" process.

In the late 1980s and early 1990s, the plethora of EDA toolset providers would have their customers believe that a particular toolset (or suite of toolsets) could automate the PCB design process. For example, EDA vendors often crowed that their toolsets could autoroute 90% of a given PCB design. Maybe so, but left unsaid was that the last 10% of the routing



1. This is a typical PCB design and fabrication flowchart with critical pain points highlighted.

was often the most difficult and time-consuming aspect of the job.

The other part of the EDA vendors' marketing mantra was that a suite of toolsets from one vendor would address all of your PCB design needs. But savvy engineers knew it was unwise to leave your PCB design to a suite of EDA toolsets.

Rather, they understood that successful design of manufacturable and reliable PCBs was best achieved using optimized point tools. Even today, the engineers who are most successful with their high-frequency, high-data-rate designs pick and choose those point tools they know will most likely realize their targeted, fully functioning, completed design.

Also, PCB tools have often been the "red-headed stepchildren" within a suite or family of EDA tools. The focus has long been, and continues to be, on components, their characteristics, and how best to leverage them within a design. The PCB tools within a given vendor's toolsets often come by way of corporate acquisitions. And, while their performance may have sufficed at the time of that acquisition, they may not keep up with emerging design challenges in the longer run.

5. It's difficult to have visibility into the fabrication process when designing a PCB.

While this statement was true in the past, there are EDA tool providers who more closely link the PCB design and manufacturing processes together. Several factors and datasets come into play during the manufacturing process that can influence the board-design process. Included among them are resin content, dimensional stability, and glass weave.

The biggest challenge is educating designers on those aspects of the fabrication process that can affect their designs. To this end, we need more synergy between the design and fabrication processes, and more sharing of information on those aspects of PCB manufacturing that can create hidden "gotchas" in the end products.

6. Multiple re-spins of PCBs are an accepted way of doing product development.

In truth, this statement is less a myth and more the "traditional" way of doing business in the PCB industry. In decades past, the only way to verify your PCB design was to fabricate a prototype for population and testing. If issues arose, that meant going back to the drawing board. This process was repeated time and again until there was a working PCB.

Even worse, sometimes a seemingly correctly designed board was manufactured in volume only to have some or even all of them fail in the final product. The result of such calamities was high NRE costs, lost time to market, and lost competitive advantages that caused profits to suffer.

Thus, the industry came to accept multiple re-spins of a board as standard operating procedure. Sadly, many OEMs still accept multiple PCB re-spins as part of the cost of doing business. This way of thinking can infect the product-development process from the start with the notion that a completely optimized board design is unachievable. As a result, the PCB development process often avoids much-needed attention or scrutiny.

7. Designing the PCBs used in today's high-frequency, high-data-rate designs has lots of "wiggle room."

The fact is that today's high-frequency, high-data-rate products have left very little wiggle room in the PCB design process, particularly in the arena of board stackup. In 112-Gb/s/

channel and 5G/mmWave applications, a slight misstep can quickly render a design inoperable or unmanufacturable.

A complete stackup design is one that includes fully integrated electromagnetic and thermomechanical simulations. A technically correct and financially optimized design approach is one that affords accurate prediction of resin starvation and glass-stop issues, and it provides board-level thermomechanical properties including coefficients of thermal expansion in all directions—X, Y, and Z axis (both above and below Tg). With that information in hand, the designer can more accurately predict the board's reliability and performance.

8. Beyond dielectric constant (Dk) and low loss (DF), there's not much insight as to how other material characteristics will affect a PCB.

While this was true in the past, more insight is available today, especially with the growth of high-frequency, high-data-rate designs. While more information is always good, the ability to put meaning behind the information becomes the crucial factor within the process.

9. The material information provided from laminate vendors is complete.

Dk and DF information has long been available from laminate vendors and, by and large, that information is reliable and reasonably complete. The challenge involves extending information beyond those two properties.

Mechanical properties as a function of construction haven't been available. In effect, each stackup presents different properties, even when built on the same material, and this can have serious implications for the manufacturability and reliability of the board. Often, material test vehicles are used to qualify a particular material, but each stackup may behave very differently from the test vehicle.

The characteristics shown in **Figure 2** go far beyond the typical Dk and DF information

Materials Properties Validation

Material Property	Symbol	Experimental Measurement	Gauss Stack Prediction
Coefficient of Thermal Expansion Z-Direction Below Tg	$\alpha (Z_1)$	~72 PPM / °C	75.57 PPM / °C
Coefficient of Thermal Expansion Z-Direction Above Tg	$\alpha (Z_2)$	~290 PPM / °C	279.57 PPM / °C
Coefficient of Thermal Expansion X-Direction and Y-Direction	$\alpha (X)$	~20 PPM / °C	20.51 PPM / °C
Young's Modulus (X-Direction)	E_x	20.56 GPa	21.565 GPa
Young's Modulus (Y-Direction)	E_y	22.06 GPa	21.637 GPa
Poisson's Ratio (xy)	v_{xy}	0.258	0.242
Poisson's Ratio (yx)	v_{yx}	0.242	0.243

It should be noted that all experimental numbers have inherent variability, as they are sensitive to setup configurations, sample preparation, and range used to extract values. Gauss Stack's predictions are well within the **measurement variability** for these test methods.

2. High-frequency, high-data-rate designs require additional laminate characteristics beyond the typical DF and Dk information provided by suppliers.

typically provided by material suppliers. They all can impact the final design, manufacturability, and long-term reliability of the PCB.

10. It doesn't require lots of training to become proficient in using EDA tools that address complex design elements such as signal integrity.

More than a few EDA tools utilized for critical issues like signal integrity have been difficult to use and require a lot of expertise to obtain the relevant data. Because these toolsets aren't employed on a day-to-day basis, it's not unusual for engineers to have to "relearn" the tool each time they undertake a new PCB design.

Getting up to speed can exceed the product-development timeline. It may also enable errors to creep into the process because the engineer using the tool is not fully familiar with its operation and doesn't know when to question critical results.

11. New or updated EDA tools all have the same construct and are based on the same internal technology.

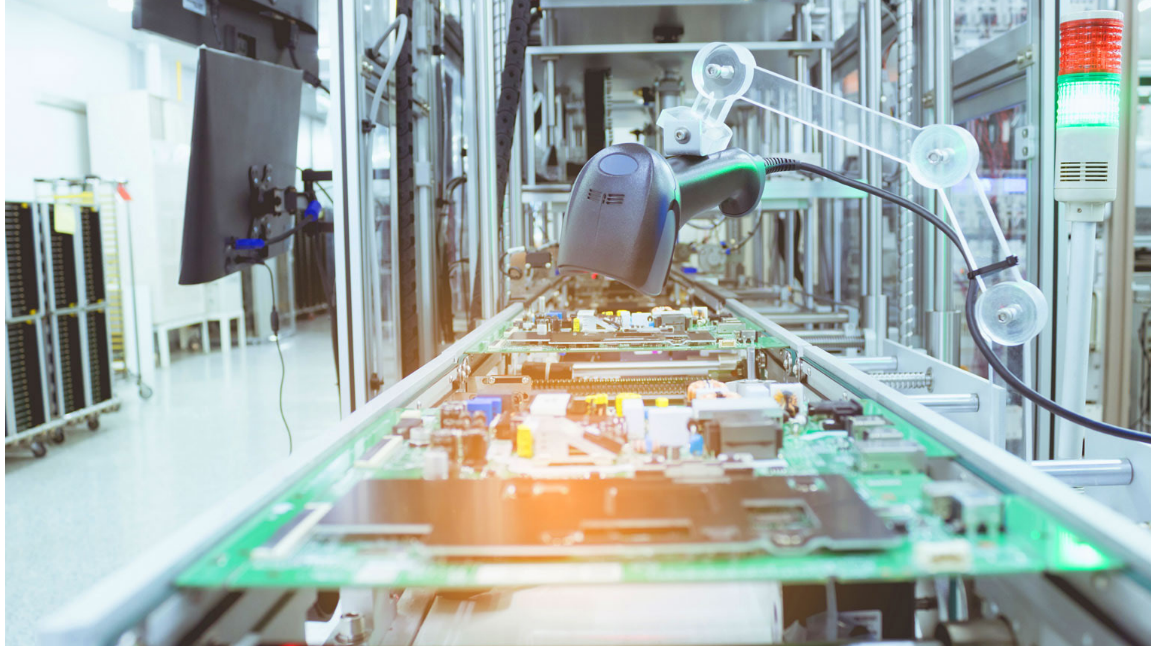
In many instances, revisions to existing toolsets or the creation of new ones has focused on adding more information, often in the form of libraries. While more information is always better, all libraries aren't created equal. Often, it's difficult to ascertain which data is good or bad.

Without the ability to manipulate data in a meaningful way or to use a new construct (like a new algorithm or simulation capability) to better leverage the vast set of information within a toolset, the benefits realized from a "revised" or "brand-new" toolset may fall far short of what's required by today's product developers. This is especially the case for high-frequency, high-data-rate products.

In addition, the ability to provide EDA design services via an online, pay-as-you-go business model can result a highly effective and reliable PCB design environment. Such approaches meet the needs of many PCB product developers who don't require a resident toolset and lack the budget to pay for a yearly subscription "per-seat" license that's the norm with many EDA toolset providers.

to view this article online, ☞ *click here*

☞ BACK TO TABLE OF CONTENTS



Dreamstime: RatchanonKlamtawee

CHAPTER 2:

Material Evidence: How Laminate Properties Impact the PCB Design Process

KELLA KNACK, Director of Strategic Marketing, *Avishtech*

For high-speed, high-frequency PCB designs, laminate evaluation and selection are critical factors in achieving a manufacturable, operable, and reliable product.

Numerous strides have been made in the tools and technologies used to design, manufacture, assemble, and test printed circuit boards (PCBs). At this point, it might be readily assumed that due to the long history of PCB fabrication, the PCB design process doesn't have any major "gotchas." But that's not the case, and they often come in the form of the physical properties and performance characteristics of the laminates used to make PCBs.

In particular, as we go up the performance curve with today's high-speed, high-frequency applications, there's a greater need to focus on how these factors impact product design, manufacturability, operability and reliability.

At the highest level, the challenges with laminates are twofold:

- Accounting for all properties and characteristics that are beyond the "basics."
- Assessing the laminate-related issues for your product and addressing them early in the design process.

This article will address these challenges by providing the upper-level guidelines for the laminate evaluation and selection process. It also highlights which properties and characteristics can impact your design based on the final end-application of your product.

A Brief History of Laminate Choices

For a very long time, PCB material choices were limited to some combination of epoxy-glass and polyimide that came to be classified as FR-4 laminates and other ANSI grades. These laminates were the workhorses of the industry, and they're still in wide use today. The performance characteristics of importance in these laminates was thermal properties and processability, and that's since shifted to low permittivity (Dk) and loss tangent (Df).

However, while FR-4 laminates were widely used even in the "dark ages" of PCB designs, some "fringe" applications—such as those found in mil-aero or defense products—specified a different laminate with little known about their long-term manufacturability, operability, or long-term reliability.

For example, when I was an engineer at Martin Marietta several years back (more than I care to admit), the PCB material of choice for the Low Altitude Navigation and Targeting Infrared for Night (LANTIRN) program was Kevlar. LANTIRN was a combination of navigation and targeting pods that were mounted underneath the wings of fighter aircraft (e.g., F-15 and F-16). Thus, the boards in them had to operate reliably in a ruggedized, harsh environment at high temperatures.

While Kevlar fit those environmental conditions, what we didn't know was that Kevlar-based boards were subject to "measling, crazing, and haloing" due to the fab process. The fabrication specification levied on our program, MIL-P-55110, prohibited these "defects." In turn, all efforts came to a screeching stop while we went through an extended QA process to verify that these cosmetic flaws would not affect the operation and reliability of the two-pod system.

These days, it's difficult to imagine any product development effort could withstand the kind of stoppage noted above. However, our industry has morphed to the point that PCB laminates are pushed to the edge to satisfy today's high-speed and high-reliability performance requirements. It's actually more accurate to state that today's PCBs and the laminates used within them have very little flexibility or "give" in terms of their operability and reliability.

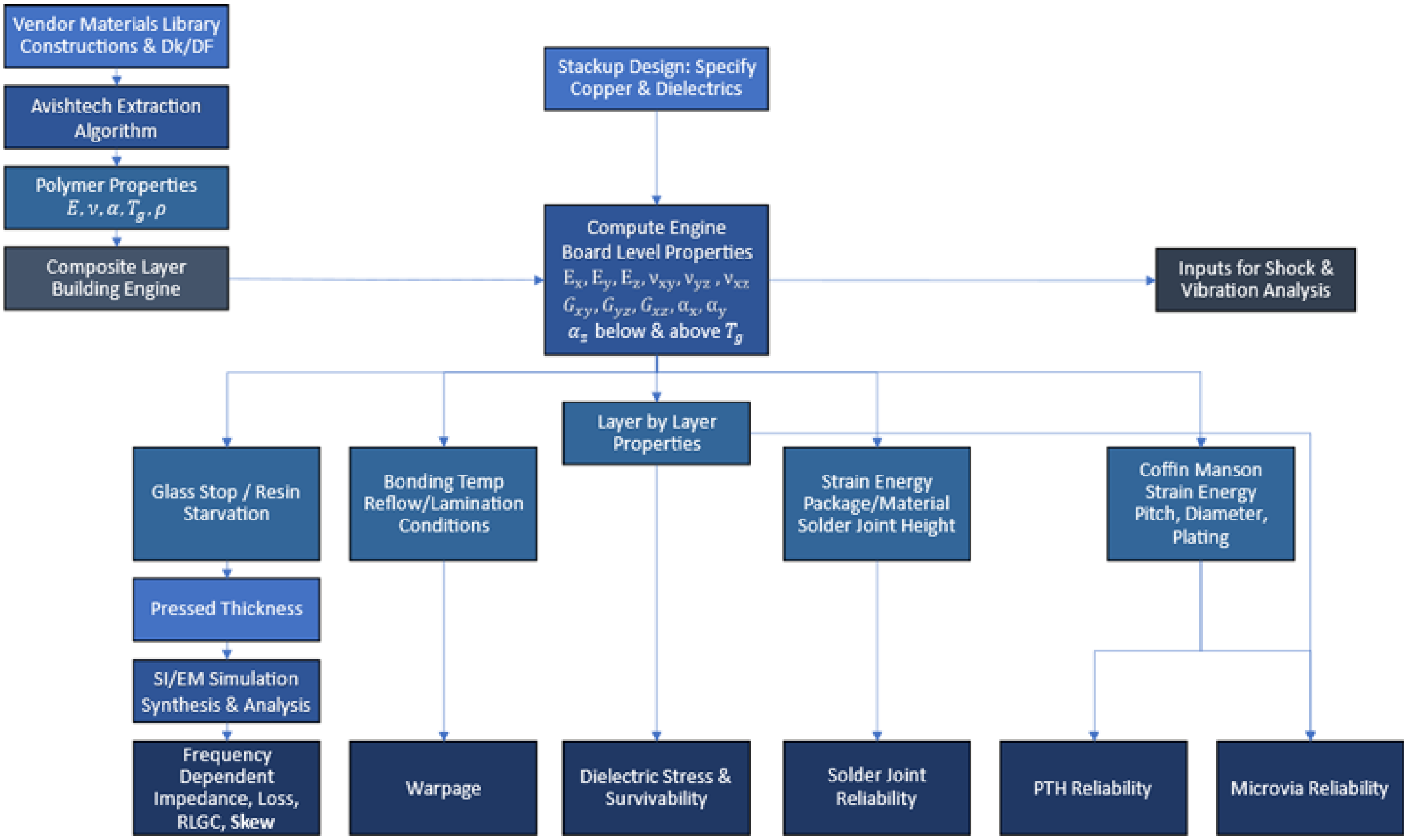
Presently, single points of failure in any aspect of the design, including the PCB and its structural components, can spell the end of a particular design, end-product, or program. And, when we discuss the importance of PCB laminates, it's necessary to consider the broader business aspects associated with their use: the need for redesign, expensive non-recurring engineering costs, multiple respins of boards, missed product windows, and reduced profit margins.

What's Important in Today's High-End and Mainstream Designs?

Beyond the fundamental attributes of Dk and Df noted above, other equally important laminate-related factors contribute to the function of a device and its long-term survivability. These include mechanical and thermal properties such as stiffness, expansion coefficients, properties of copper, and copper roughness. These properties not only the address the processability of the materials, but also the field reliability of the product. The **figure** shows the workflow that goes into the laminate evaluation and selection process.

In many cases, it's the end-application and performance of the products that dictate which laminate-related properties are the most crucial. For high-end boards found in routers and servers, the speed of processing the data is the key aspect. For handheld devices, in addition to their computing capabilities, other factors of concern include their ability to withstand multiple drops, tolerate vibrations, or withstand high heat such as that found in a car's interior. It's important to keep in mind that all of these performance requirements directly relate to the intrinsic characteristics and properties of the laminates from which they're constructed.

While all of the foregoing factors play a role during the material evaluation process, the



This is the workflow that goes into the laminate evaluation and selection process, along with the design-specific elements that will have a bearing on that process.

challenge is that not all laminate suppliers do the investigation required to determine whether their laminates will support those factors.

Types of Tests Being Done

The types of tests that most laminate suppliers will perform are CAF testing and thermal testing. CAF (conductive anodic filament formation) is a process wherein the conductive chemistries are transported across a non-metallic substrate under the influence of an applied electric field. It's influenced by electric field strength, temperature (including soldering temperatures), humidity, laminate material, and the presence of manufacturing defects. As noted later in this article, CAF is often responsible for field failures in humid environments.

At the behest of the automotive manufacturers, some laminate suppliers also will conduct thermal cycle testing. In these situations, an OEM automotive supplier will provide the information to the laminate provider and state "subject the laminate to alternating thermal cycling conditions simulating an Arizona summer to a Canadian winter." It's important to note that only a few laminate suppliers will do this kind of cycling.

The Influential Elements Within Laminates

Years back, when "high-speed" end-applications were operating at a few gigahertz, lots of latitude was given in terms of the impact laminate properties and characteristics had on final products. In contrast, now, as we push to 112 Gb/s in current designs, there's not much leeway at all, especially for the aforementioned Dk and Df characteristics.

In addition, the loss budgets for these high-speed devices are extremely low and, in most cases, they're beyond the capabilities of both the laminate and the conductive, which is copper. As a result, the choice of materials and the copper widths become very critical.

With early mobile devices, many instances of bad designs made it to market. And, a lot of the "failures," such as glass shattering, were left to the next rev of the design to ameliorate the problem.

In mission-critical designs, there's no room for the error that can lead to the failure of an entire system, and it's not possible to wait for the next product iteration to address the problem. Thus, these designs must be robust in terms of their ability to perform multiple functions, reliably, over a long period of time. For such designs, it's the interplay of the thermal/mechanical properties like expansion, Tg, modulus, shear properties, and fracture toughness that has to be taken into account.

The foregoing describes the key elements in getting a board manufactured. However, the PCB manufacturing process itself also is very demanding and grueling. In almost all instances, it takes materials well above their elastic regions.

The processes of lamination, drilling, and reflow place stresses and strains on the laminate that put them at their breaking point. And, ultimately, very small deviations can lead to the failure of a design. We don't see 6 sigma kinds of numbers in our industry and yields tend to be between 70% to 90%. If the laminate in a board can't survive these processes, you're right at the very end of what is achievable, and designs can fall off.

Why You Can't Rely on Just One Data Point for Your Design

Just as we examine the tradeoffs in a PCB design between manufacturing, assembly, reliability, test, and cost, there's a need to examine the properties within laminates to see how well they play with each other. That's what's been at the core in the development of our Gauss Stack product. It enables you to look at the impact of these properties for a given material, in a given design, and determine whether they will work well together or if everything needs to go in another direction.

With certain high-speed designs, which are HDI types, multiple, sequential laminations are required. So, in addition to the low Dk and Df, low levels of CTE and high adhesion are needed as well, not to speak of thermal robustness. However, when you add something in the laminate system to have low CTE, it will adversely affect the dielectric properties and adhesion.

The final laminate choice must strike the optimal balance between product performance, the ability to survive field conditions, and the inclusion of low CTE. It's a delicate balance to arrive at a good design that fulfills all operation requirements and performs optimally in field operations.

As noted earlier, the challenge is that laminates are composites comprised of polymers and reinforcements such as glass and fillers. That's why having just one data point from a datasheet isn't sufficient for most designs.

When properties are limited to Dk and Df, the data on mechanical properties (modulus, expansion, and CTEs) aren't available. Whether you're building high-layer-count boards or those that are four- or six-layer, you can't just rely on a single data point for your design.

To further complicate matters, for a number of years, and somewhat still today, there are instances where the data in a datasheet is wrong. Because product developers may not have experience with all of the properties and characteristics of a given laminate, they can't determine if the information provided is questionable. Fortunately, we've seen this problem be fairly well eradicated because laminate suppliers have been called out on supplying incomplete or erroneous data.

Going in the opposite direction, some laminate suppliers will overstate their Dk and Df numbers. Then, when product developers simulate their numbers, they will achieve better results in practice. This actually adversely affects the laminate supplier, but as long as they are getting sales, they tend not to worry about the errors in their data.

How Product Development Efforts Factor into the Equation

A key challenge in understanding the interaction between laminate properties comes from how interactions are handled during the product development process.

For years, our industry has been characterized by people working in silos even within OEM groups. Designs are handed over from the electrical to the mechanical designers. And, when these groups only rely on the information on material datasheets, it often diminishes the needed interplay and multidisciplinary approach to product design.

Taking this one step further, those in charge of designing the product don't understand what the laminate properties and characteristics can do to their product because they don't see this impact at their level.

This has been the source of practices such as rework and respin, which are embedded in the design process and typically considered to be part of the price of "doing business." But what frequently happens is that product launches can get delayed, stymied, or shelved because the laminate information wasn't correctly considered during the design process.

Another frequent occurrence is that software and modeling tools tend to over predict the performance for a specific laminate. This is especially true in the case of loss. Then, when failures occur, the product developers will ask the laminate suppliers why their laminate isn't working, while the laminate supplier will ask why they weren't involved in the evaluation process during the earliest phases of the design effort.

Part of the problem is that much of the product development activity is driven by purchasing departments. This is best described as the "cost vs. price" business model. Laminate selected at the lowest cost can result in the highest price when it's found that the end product is flawed or doesn't work at all.

One of the best ways to ensure that the product you're designing is manufacturable, operable, and reliable across the entire product lifecycle is to involve everyone in the design process. The laminate supplier, the product developer, and the manufacturer are equally important. In some instances, it's necessary to involve the assembly and test groups as well.

Verification vs. Validation

Though several good verification software products are available to the industry, the only way to know for sure that your product will work as designed and manufactured is to do real-world validation.

Within our Gauss software, we enable our customers to screen several materials and their expanded properties and characteristics. Then they move to the next stage, which is to do validation on two or three prototypes. At the end of the day, real-world validation is the only way to ensure that your design will work as designed and manufactured.

The foregoing practice isn't without its challenges. With a lot of secrecy surrounding new builds, product developers don't want to send their design to a fabricator that may significantly compromise that secrecy (as has happened in some offshore facilities). Also, real-world validation is expensive. Fabricators can ask for and get hundreds of thousands of dollars to validate a design.

Doing the validation computationally is great but may not always be available as a solution. On top of that, the level of technology involved is another problem. For instance, if you want to build a board with microvias or stacked microvias, there are only a few manufacturing facilities that can do that kind of manufacturing.

In addition, such complex high-end designs only account for 6% to 7% of the boards being built for the entire PCB market. As a result, people will do validation builds like volume runs and spend between hundreds of thousands and one million dollars just to get their product validated.

Unfortunately, if you haven't taken the steps necessary to ensure that your design is manufacturable, you end up going back to square one. Fine-tuning the verification process wherein you narrow the options down to two or three prototypes is the most effective way to arrive at a real-world, validated design.

Other Challenges

Taking all of the foregoing into account may not be enough when it comes to material properties and characteristics. One of the biggest impacts on a type of laminate is aging of the material. When the lead-free conversion happened, a great deal of material was brought onto the market almost immediately. Only later was it discerned that the material didn't work too well with humidity. CAF is the classic example of what can happen over time and lead to field failures.

Bottom line, something that absorbs moisture will degrade over time. You can do signal-integrity work with pristine dry material that works fine in a dry environment. Then when the product is running in hot, humid conditions in some other environment, the material will degrade over time and the product won't perform as specified or perform at all.

Again, taking into account all of a design's performance characteristics in terms of its inherent operation, as well as the environment into which it will be deployed, goes a long way to ensuring that the product will perform correctly.

Why the Stackup Design is So Important

One of the key efforts involved in any successful PCB design is creating a really good stackup. Unfortunately, it's necessity and exactitude are often not given near enough consideration. Seemingly benign changes to the prepreg layer or the stackup, or replacing one laminate with another that has a higher or lower resin content, can lead to warpage, make the board unmanufacturable, or lead to CAF problems.

Not long ago, we witnessed a design where the glass type in the laminate was changed, resulting in massive voids. The outcome was that the entire product was shelved after two years of development effort. It was a disastrous result and could have been prevented if someone had addressed it at the onset of the design process.

Unknowingly, product developers may think that adding or removing a layer or replacing 1 oz. copper with 2 oz. copper won't make any difference—and then they're left to wonder why their boards are delaminating. Paying proper attention to the stackup process, understanding how specific materials properties will affect design, and taking as many downstream processes into account early on in the design effort goes a long way to achieving a properly designed and built product that's also profitable.

What Factors Should You Consider in Your Design?

While the impact of laminate selection is based on the end-application of your product, here are some of the main laminate-related issues that you will want to avoid in your final product. Technology now exists to address them proactively during the creation of the stack-up design.

- Glass stop and resin starvation
- Filler damming
- Plated-through-hole reliability
- Microvia reliability
- Solder-joint reliability
- Dielectric stress
- PCB warpage

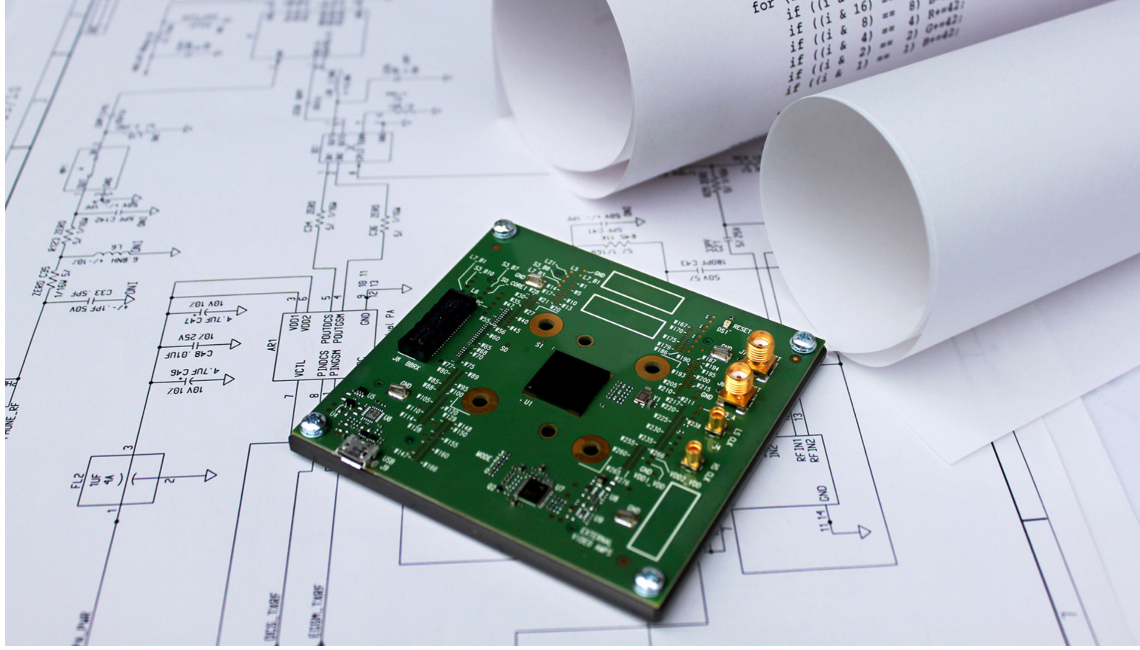
Summary

We've come a long way when it comes PCB design, manufacturability, and operability. For those of us who have been in the industry a long time, each new product development challenge creates lots of head scratching as we determine if the end product can be developed using traditional design approaches and laminate materials. Nonetheless, we do so.

Then the next product innovation comes along, and we start to wonder again how we will overcome the technology hurdles required to build it. That's the challenge and it's what makes all of us get up in the morning.

to view this article online, *click here*

BACK TO TABLE OF CONTENTS



Dreamstime: Outline205

CHAPTER 3:

IPC Standards for PCBs: What Are They and Why Do They Matter?

SURESH PATEL, Sales Engineer, *Mer-Mar Electronics*

This article shares some PCB assembly standards that one should follow to ensure a successful PCBA project, as well as revealing some benefits of the IPC standards.

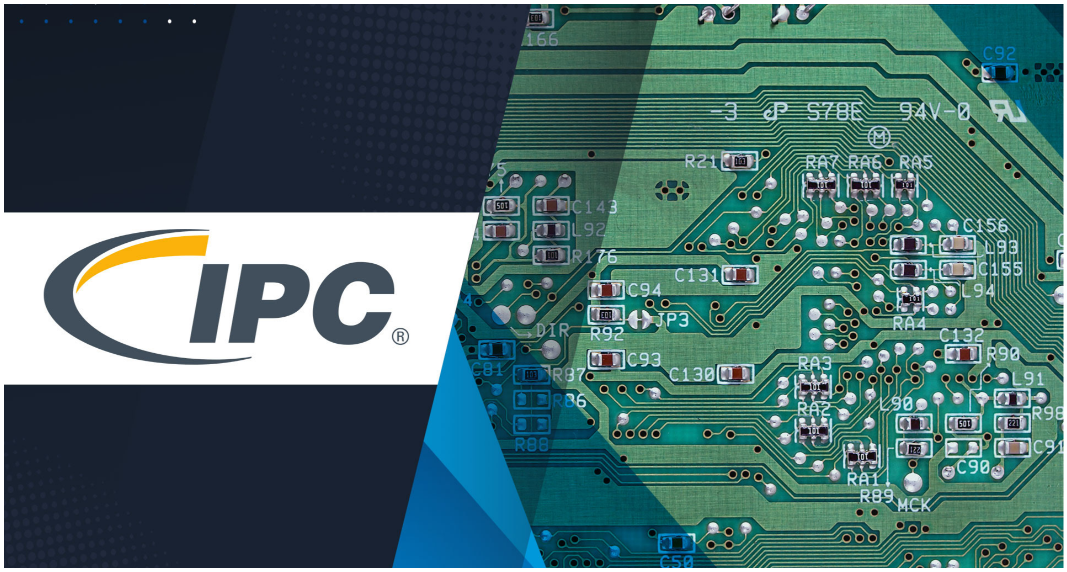
The Institute for Printed Circuits (IPC) is as known as the Association Connecting Electronics Industry. It's a trade association for the electronic interconnection industry that authorizes high-quality electronics by developing the standards to drive the success of the global electronics industry.

IPC, initially established by six PCB manufacturers in 1957, now consists of more than 3,000 companies that design, manufacture, and assemble PCBs for applications ranging from consumer electronics and automotive, to telecom and healthcare. IPC members are from different facets of the electronics industry such as original equipment manufacturers (OEMs), PCB manufacturers, PCB assembly service providers, and suppliers.

IPC works with the electronic industry to identify industry challenges and collaborates with its members in providing suitable solutions. It also gathers the latest data related to the electronics manufacturing industry and outlines appropriate standards for best practices.

The standards developed by the IPC are applicable in every stage of the PCB production, from design to manufacturing to final assembly. There are acceptability standards at various stages of development to ensure a high-quality end-product. **Some key takeaways of the IPC standards include:**

- The terms and definitions used in the standards are listed in the IPC-T-50 document.
- The standards for the file formats, product documentation, and design software are listed in the IPC-2581 and IPC-2610 series.
- The design and land patterns are covered in standards like IPC-2221, 2222, 2223, and 2226 as well as IPC-7351.
- Substrates and base materials for PCBs are expected to meet the standards mentioned in IPC- 4101, 4103, 4104, 4202, 4203, and 4204.
- Printed-board quality and reliability standards are outlined in IPC-6011, 6012, 6013,



6017, and 6018.

- Printed-board acceptability standards are described in IPC-A-600.
- Requirement criteria for soldered electronic assemblies are listed under IPC-J-STD-001, IPC-HDBK-001, and IPC-AJ-820.
- Acceptability standards for electronic assemblies are outlined under IPC-A-601.
- Cable and wire harness assemblies have the requirements and acceptance standards elaborated in IPC/WHMA-A-620, IPC-D-620, and IPC-HDBK-620.
- The acceptability standards for manufacturing, inspection, and testing of electronic enclosures are covered under IPC-A-630 and IPC-HDBK-630.
- There are standards for advanced packaging, storage, and handling, too.

Some of the IPC standards relevant to PCB design and manufacturing include:

- IPC-1752A establishes a standard reporting format for material declaration data exchanged among the designers, distributors, and manufacturers. It also supports reporting of bulk materials, components, assemblies, and products.
- IPC-2152 defines the current carrying requirements in copper traces and planes in a PCB design.
- IPC-2221 prescribes some generic design and performance requirements in PCBs and other methods of component mounting or interconnecting structures.
- IPC-2581 specifies a generic requirement for sharing information among supply-chain participants. It provides a standard format for exchanging design data that ensures reliable production outcomes.
- IPC-4101 covers requirements for a wide range of PCB laminates or prepregs used to build stackups.
- IPC-4761 reflects the design guidelines for via protection to ensure dependability, manufacturability, and quality.
- The IPC-A-600 series defines the acceptability criteria for printed circuit boards, as well as the three standard IPC classes for various PCB assemblies:
    - *Class 1:* This includes consumer electronics and general products that are assembled at the cheapest possible price and expected to function for a set duration.

  - *Class 2:* Under this category, products like laptops or home appliances are included. They are service-oriented products with dependability and an extended lifetime.
  - *Class 3:* Critical performance electronics products like medical equipment and aerospace apparatus fit this category. They are expected to operate nonstop in harsh environments demanding very high reliability and robustness. IPC-7351 specifies the footprint and land-pattern requirements for surface-mount (SMT) components. SMT PCB assembly providers follow these specifications while designing the land patterns to ensure solderability and reliable electrical connections.

IPC standards are incorporated in design guidelines and can be verified using effective checklists throughout the development process. Customers now prefer products that guarantee user safety along with high performance. Hence, adhering to the IPC standards shall certify a PCB product to be of excellent quality.

Benefits Derived from IPC Standards

IPC standards bring several benefits to the electronics industry, such as:

- *Enhanced product value and consistency*: Strictly following the IPC standards throughout the development and manufacturing process leads to a high-quality product, further improving product reliability and customer satisfaction. IPC standards hence contribute to the consistency and overall product value in the market.
- *Better communication among contributors*: When all of the involved members follow the same IPC standards, it becomes easy to communicate between the groups. The common jargon assists in the clear interpretation of set standards for each deliverable and avoids any ambiguity during communication.
- *Improved brand identity in the market*: Following IPC standards helps build your brand value and sets a good reputation in the market. It indicates the commitment of the manufacturer toward product quality. This can bring in more collaboration and opportunities to expand your business.
- *Lower cost*: Adhering to the IPC standards can help a manufacturer improve supply-chain efficiency, which can further reduce the resource requirements. And it will substantially lower production cost.
- *Seamless workflow*: When all team members diligently follow IPC standards, it also significantly reduces the possibility of faults and delays. This leads to a smooth flow in the development, production, and assembly lines of a PCB product.

The above advantages justify the importance of IPC standards for PCBs, and they do matter for any PCB manufacturer and contract manufacturer (CM).

An experienced CM will not just focus on a few details related to assembly; rather, they integrate the whole process with a strong foundation based on the complete set of IPC standards. As a result, the CM can identify even a small PCB manufacturing defect in the assembly line and improve productivity. Collaborating with an experienced CM offers another route toward building a high-quality PCB product.

to view this article online, *click here*

BACK TO TABLE OF CONTENTS

CHAPTER 4:

Characterization of Epoxy Prepregs by DSC

TA INSTRUMENTS

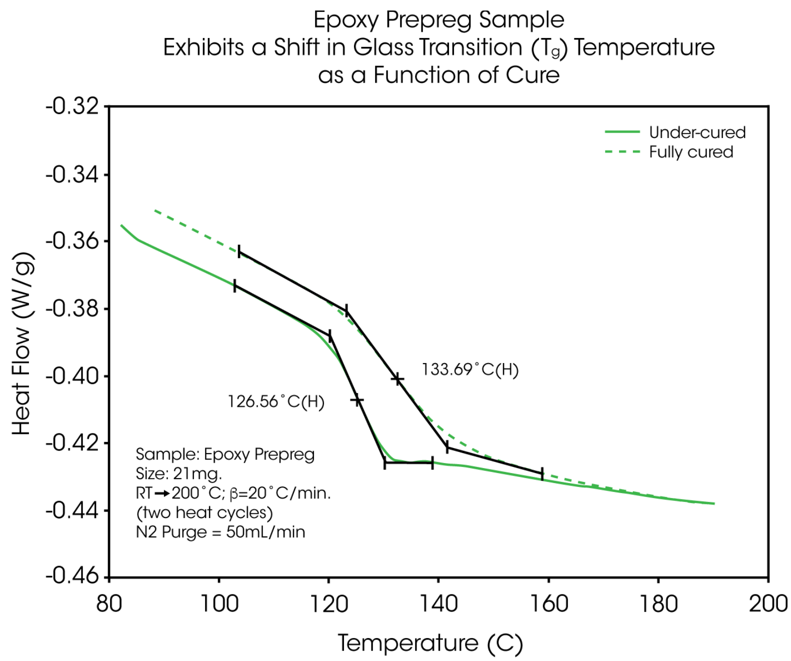


Figure 1. Heat flow curves of epoxy prepreg sample measured by DSC

Differential scanning calorimetry (DSC) measures the temperatures and heat flows associated with transitions in materials as a function of temperature or time in a controlled atmosphere. This technique provides quantitative and qualitative information about physical and chemical changes that involve endothermic or exothermic processes, or changes in heat capacity.

A TA Instruments™ differential scanning calorimeter was used to analyze an epoxy prepreg sample. A 21 mg sample was heated from room temperature to 200 °C at a rate of 20 °C/min. Two heat cycles were used under a nitrogen purge.

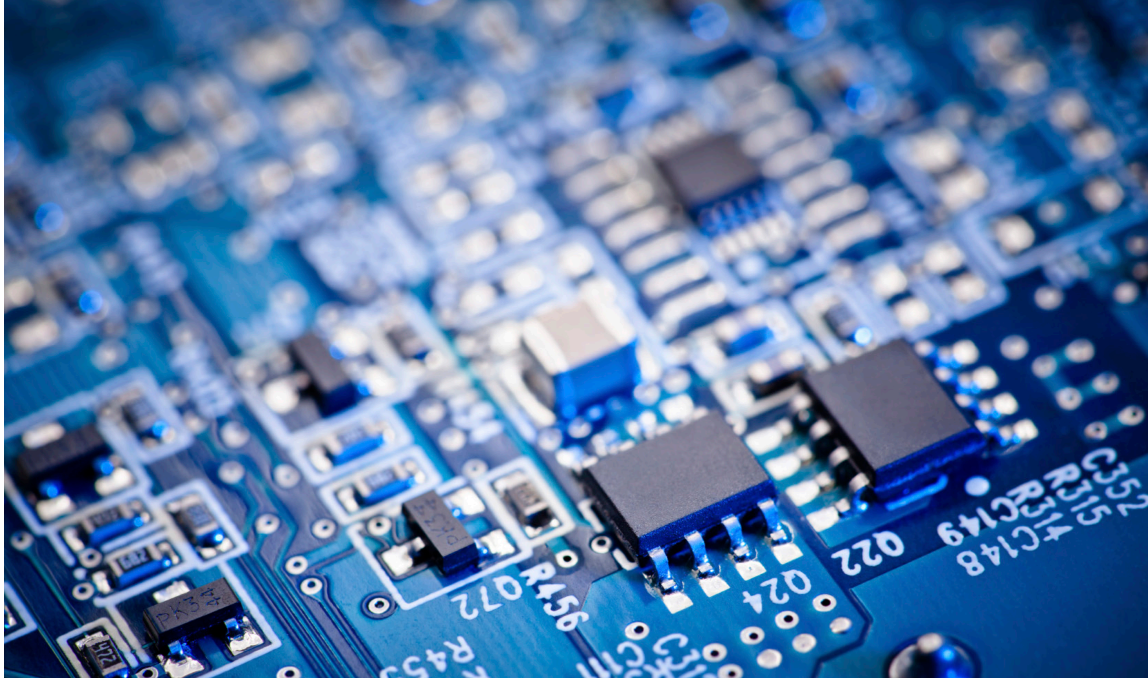
The thermal curve in **Figure 1** shows two heat cycles on an epoxy prepreg used to manufacture printed circuit boards. DSC easily detects a shift in the glass transition (T_g) temperature indicating the sample, as received, was not fully cured. The undercured sample could affect assembly operations (i.e. hole drilling). This example clearly shows how the TA Instruments differential scanning calorimeters can be used to characterize materials in both a research and quality control setting.

Click here to download the printable version of this application note.

Contact us to learn more about our instrumentation and how it can benefit your research.

to view this article online, *click here*

BACK TO TABLE OF CONTENTS



CHAPTER 5:

Simultaneous Rheology-Dielectric Measurements of Epoxy Curing

TIANHONG TERRI CHEN, PhD, Senior Applications Support Scientist, *TA Instruments*.

Abstract

When using an epoxy-based material in electronic industry, it must meet the requirement of specific dielectric and rheological performance. TA Instruments provides a dielectric accessory (DETA), which can be attached to any TA Instruments rotational rheometers. This allows a simultaneous rheological and dielectric measurement performed with one sample loading and in the same environmental temperature profile. The rheometer is a powerful instrument that can measure the viscoelastic property change of the sample during curing. The dielectric LCR meter is used to monitoring the changes of the mobility of the ions and dipoles in the material. In this study, we discuss in detail on how to properly design an appropriate test protocol for this kind of simultaneous measurement.

Introduction

Epoxies and epoxy-based composites are preferred insulating materials for many electrical applications, especially printed circuit boards, adhesives, GIS spacers, generator ground wall insulation system and cast resin transformers etc. [1-3]. These materials exhibit excellent mechanical strength, ideal electrical property, and good chemical and heat stability. A quantitative measurement of the dielectric and mechanical properties is critical for guiding formulation and manufacturing of these kinds of products.

The mechanical performance of the epoxy-based products can be quantitatively evaluated using rheological analysis. Rheology is a powerful tool, which measures viscosity and viscoelasticity of a material. For characterizing the curing of an epoxy, a dynamic oscillatory test is commonly used either isothermally or with a programmed temperature profile, which mimic the curing conditions in manufacturing. In an oscillatory test, sample moduli (G*, G'

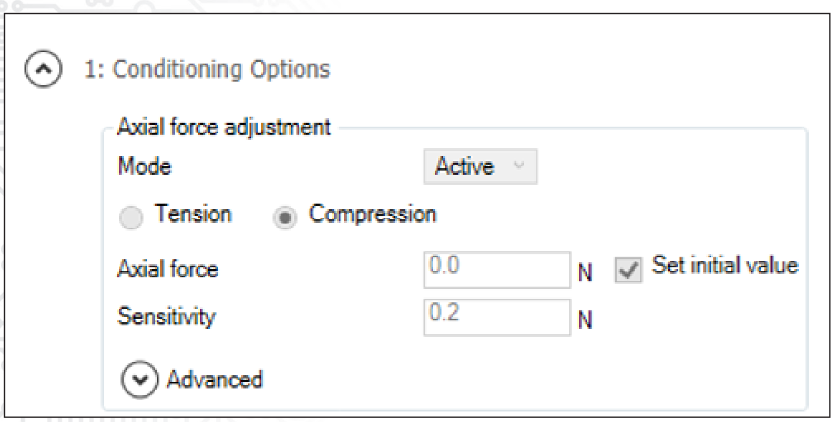
Figure 1. TA Instruments Discovery Hybrid Rheometer with the Dielectric accessory

and G”), complex viscosity (η*) and damping factor (tan delta) are recorded as a function of the mechanical test frequencies and with a small shear amplitude that is within the linear viscoelastic region of the sample. The rheological measurements help better understanding of the physical property change throughout curing. The modulus measurement for a fully cured product also provides quantitative evaluation to the performance of the final products.

Dielectric analysis is the study of the mobility of dipoles or ions in a material [4]. In a dielectric measurement, two fundamental electrical characteristics are reported: capacitance and conductance. The capacitance represents the material’s ability to store electrical energy. The conductance represents both the material’s ability to transfer electrical charge and the loss of electric energy through relaxation processes. These properties allow probing the chemistry and molecular relaxation in materials such as epoxies. It is a more sensitive technique than DMA to probe local motions along the polymer chain since the polar bonds are affected by the electrical field. Epoxy resins are good electric insulators due to low concentration of free charge carriers. Their dielectric measurements can be carried out using a LCR meter under a certain temperature profile and with a wide range of dielectric frequencies. During the measurements, all dielectric parameters such as capacitance (C), conductance (G), storage permittivity (ε’), loss factor (ε”) and the loss tangent (tan δDE) are recorded as a function of time, temperature and dielectric frequency.

TA Instruments provide a dielectric testing accessory (DETA), which can be used in combination with general rheological measurement. **Figure 1** shows a picture of this dielectric accessory attached to a Discover Hybrid Rheometer. This system allows a simultaneous measurement of rheological and dielectric properties with one single sample loading and in one test.

In this study, we elaborate in detail on how to properly setup a simultaneous rheology-dielectric measurement to monitoring an epoxy curing process.



Experimental

One commercially available 60-minute epoxy was used as an example sample for this analysis. The dielectric accessory (Keysight LCR meter E4980A) was attached to a TA Instruments Discovery Hybrid

Rheometer with the environment test chamber. A pair of 25 mm aluminum disposable parallel plates was used for the test. The two-part epoxy sample was mixed 50/50 at ambient temperature and then loaded immediately to the plate set at the initial test temperature.

During the test, the axial force control was set at 0 ± 0.2 N to compensate for sample shrinkage during curing.

The axial force control was set at 0 ± 0.2 N to compensate for sample expansion as temperature increases and shrinkage as sample cures.

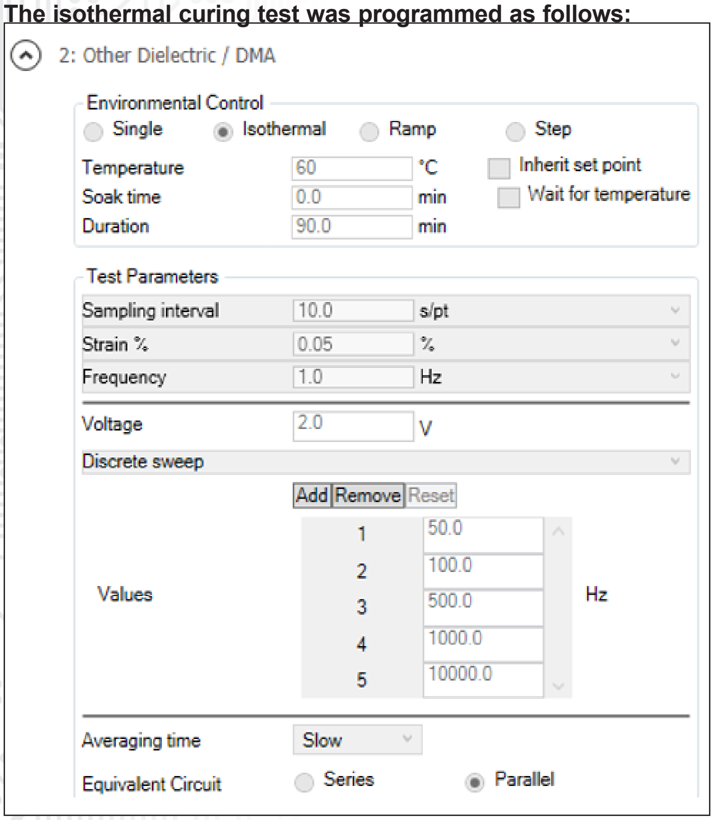
The dielectric frequencies were selected at 50 Hz, 100 Hz, 500 Hz, 1000 Hz and 10000 Hz, respectively. Since epoxy material has high impedance, the measurement voltage was set at 2 V and the equivalent circuit was under parallel.

Results and Discussions

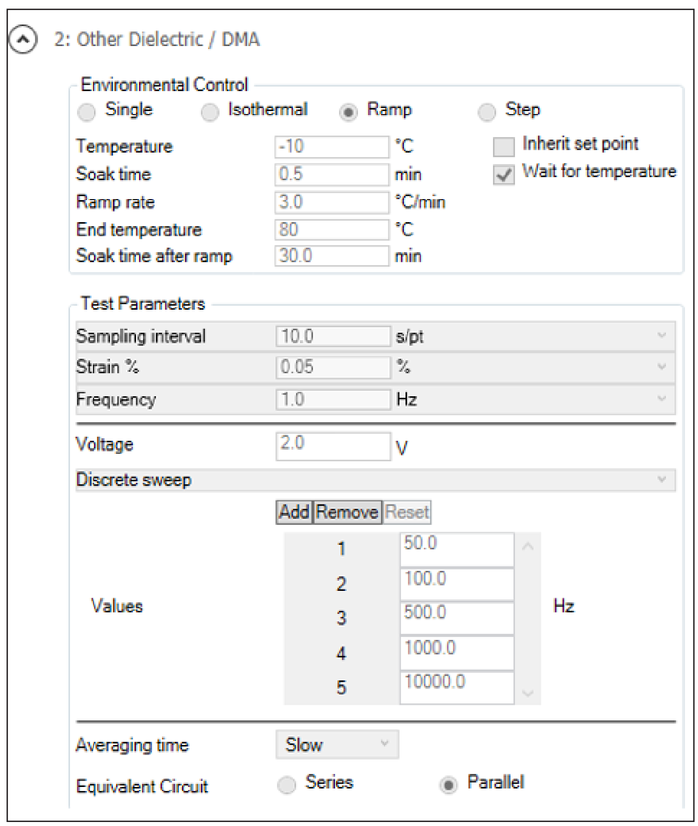
During the isothermal testing at 60 °C, the rheometer applies an oscillatory shear deformation to the sample at a mechanical frequency of 1 Hz (6.28 rad/s), then measures the sample response in stress. This mechanical testing reports sample moduli (G' and G"), complex viscosity (η*) and tan delta as a function of curing time. **Figure 2a** shows the rheological testing results. The gelation time (i.e. the G'/G" crossover time) of this epoxy sample is observed at 11.8 minute. After 1 hour, both moduli and complex viscosity curves reach to a plateau, which indicate that the curing is mostly completed.

Simultaneously the LCR meter applies an AC current at 5 separate dielectric frequencies (i.e. 50 Hz, 100 Hz, 500 Hz, 1000 Hz and 10000 Hz, respectively) (**Figure 2b**). At the begin-

The isothermal curing test was programmed as follows:



The temperature ramp curing test was programmed as follows:



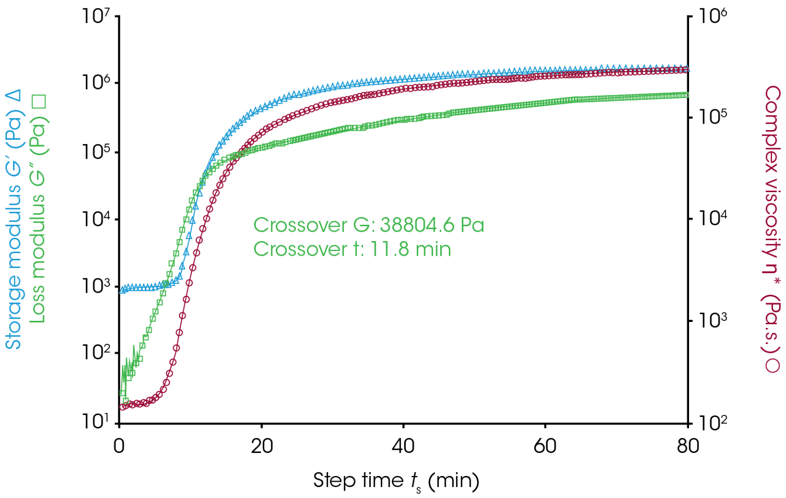


Figure 2 (a). Isothermal curing at 60 °C – rheological results

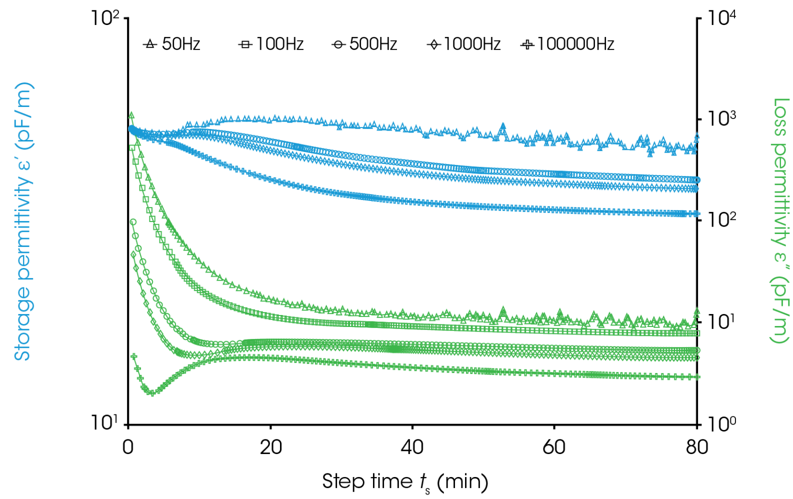


Figure 2 (b). Isothermal curing at 60 °C – dielectric results

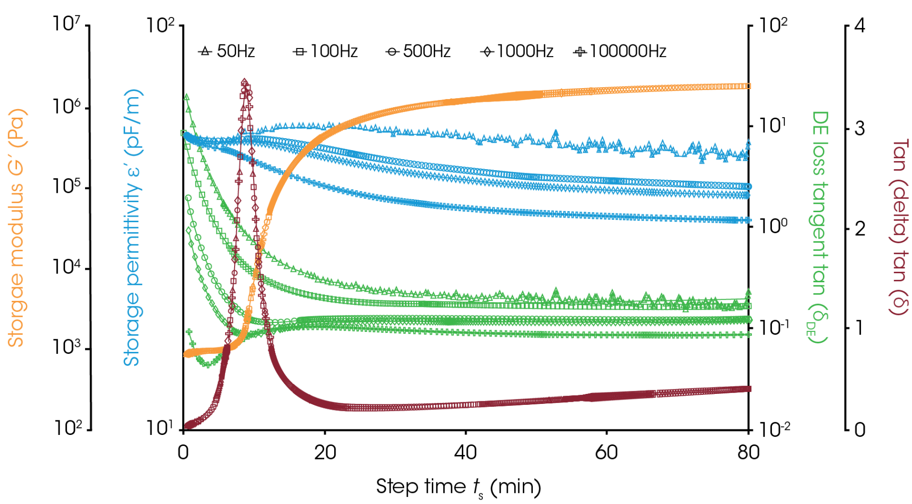


Figure 2 (c). Isothermal curing at 60°C – results overlay

Figure 2: Simultaneous rheology-dielectric measurement of an isothermal curing of a 60-minute epoxy at 60 °C. (a) Rheological results; (b) Dielectric results; and (c) Overlay of both rheological and dielectric parameters.

ning of the cure, the dipoles and ions move easily and align with the field as polarity changes owing to the low sample moduli and viscosity. The storage permittivity and loss factor are consequently high. As the cure proceeds and larger molecules are formed, the mobility of the dipoles decreases, and the relaxation time increases. The sample becomes more viscous and the resistivity (1/conductance) to the free ions increases. Therefore, both the storage permittivity and the loss factor decrease. **Figure 2c** shows the overlay plot of rheological and dielectric measurement results. This plot provides a good correlation between the rheological property and the dielectric property changes through curing process. The results show that after the sample gels, the mechanical strength increases, while the mobility of the dipoles and ions decreases.

A quantitative dielectric measurement of the uncured resin helps to guide epoxy formulation, and a quantitative dielectric measurement of the final cured material provides good evaluation of the product performance. The rheometer-dielectric system can perform individual dielectric analysis for the uncured and cured samples. **Figure 3** shows the overlay dielectric frequency sweep test results of the uncured and fully cured epoxy. After the sample is fully cured, the mobility of the dipoles is highly restricted, so both permittivity and loss factor are lower compare to the sample that is uncured.

In addition to performing isothermal analysis, the rheometer-dielectric system can also monitor the curing process with a certain temperature profile. **Figure 4** shows the results of a temperature ramp followed by isothermal curing profile. The rheological data shows that the sample is liquid-like at the beginning with G” greater than G’. Both moduli and complex viscosity decrease as temperature increases. Simultaneously, the increase of loss factor

Figure 3. Dielectric frequency sweeps of epoxy sample before and after curing

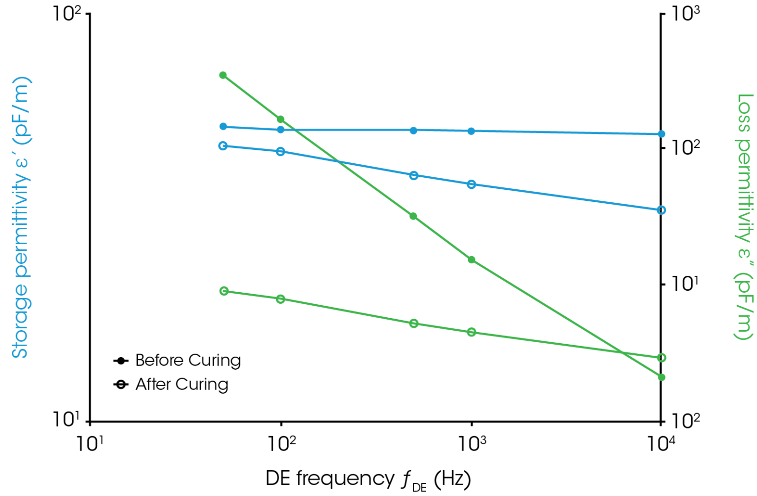


Figure 4: Simultaneous rheology-dielectric measurement of a 60-minute epoxy curing with a temperature ramp and hold profile.(a) rheological results; and (b) dielectric results versus viscosity.

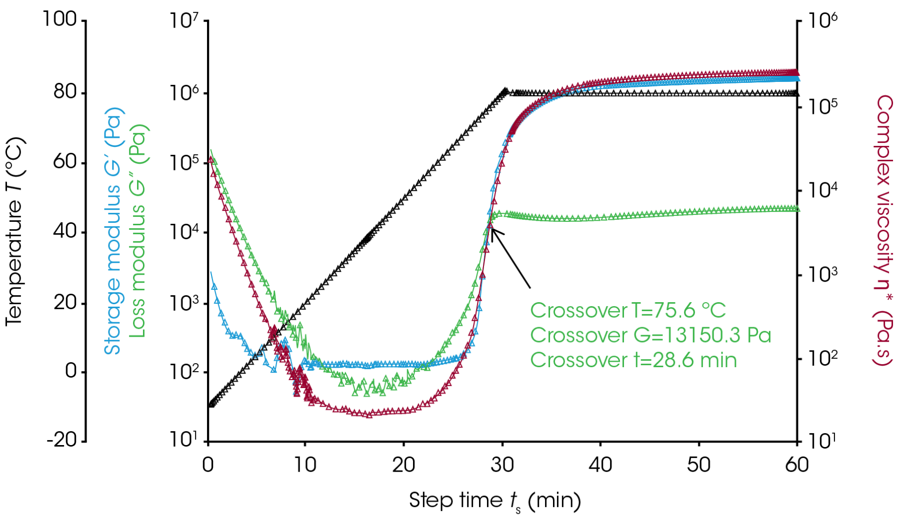


Figure 4 (a). Temperature ramp curing – rheological results

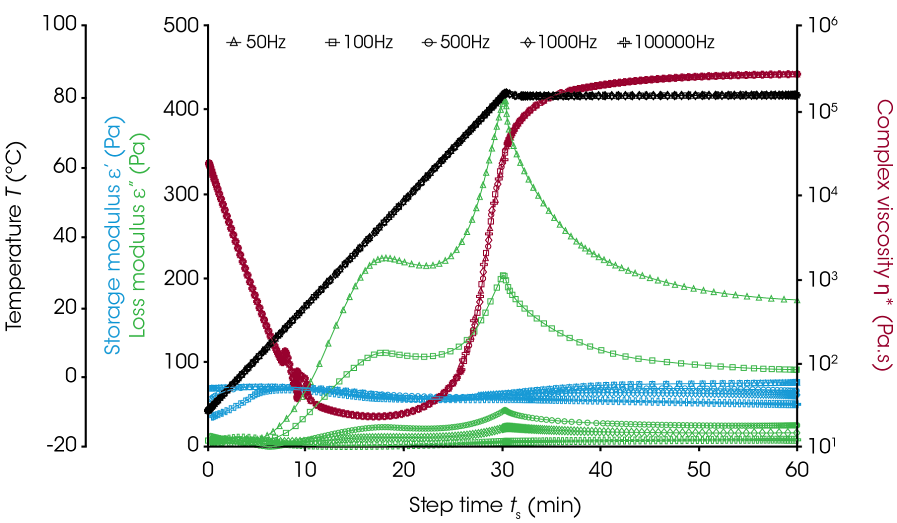


Figure 4 (b). Temperature ramp curing – dielectric results vs viscosity

shows the mobility of the ions and dipoles are higher with increasing temperature. The minimum viscosity is observed at about 40 °C, at which temperature the loss factor also shows a peak indicating a high molecular mobility. Above this temperature, curing starts, both moduli and complex viscosity increase rapidly with further increase of temperature. In the meantime, the mobility of the molecules is affected by both temperature increase, which leads to an increase in mobility, and viscosity increase, which leads to a decrease in mobility. The gelation, which is the G'/G" crossover is occurred after 28.6 minutes of reaction, and at temperature of 75.6 °C. Beyond gelation, sample becomes solid-like (i.e. G' is greater than G"), and the mobility of the ions and diploes are more and more restricted. As the test transition to the isothermal stage, the loss factor (ε") shows an immediate sharp decrease, because at this stage, the molecular mobility is only affected by the vitrification of the material.

Conclusion

The TA Instruments dielectric testing accessory (DETA) for rheometers can be used to perform simultaneous rheological and dielectric measurements. It can be used to quantitatively monitor epoxy curing with one sample loading and in one single test protocol. The rheometer conducts shear measurement and reports mechanical properties such as modulus and viscosity, while the LCR meter performs simultaneous dielectric measurements and monitor molecular mobility and relaxation. The results provide good correlation to the material's structure-property relationship.

References

1. W. Jilani, N. Mzabi, O. Gallot-Lavall´ee, N. Fourati, C. Zerrouki, R. Zerrouki and H. Guermazi, Dielectric relaxations investigation of a synthesized epoxy resin polymer, Eur. Phys. J. Plus 130: 76, (2015)
2. S. Singha and M. J. Thomas, Dielectric Properties of Epoxy Nanocomposites, IEEE Transactions on Dielectrics and Electrical Insulation, Vol. 15, No. 1; February (2008)
3. M. J. Kadhim, A. K. Abdullah, I. A. Al-Ajaj, and A. S. Khalil, Dielectric Properties of Epoxy/Al2O3 Nanocomposites, International Journal of Application or Innovation in Engineering & Management (IJAIEM), Volume 3, Issue 1, January (2014)
4. Aly Franck, Dielectric Characterization, TA Instruments applications notes APN032

Click here to download the printable version of this application note.

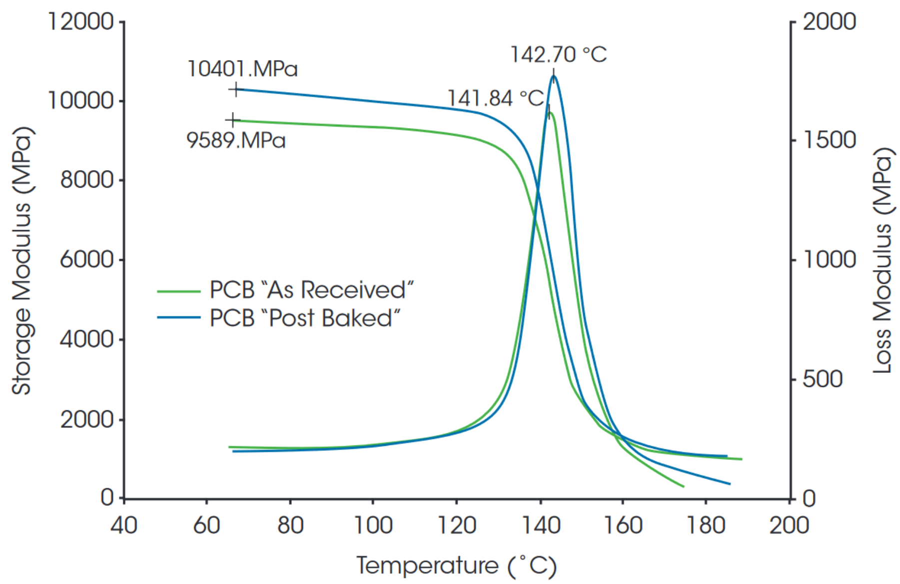
to view this article online, ☞ *click here*

☞ BACK TO TABLE OF CONTENTS

CHAPTER 6:

Characterization of Printed CircuitBoard Materials by DMA

TA INSTRUMENTS



DMA Experimental Conditions:
Clamp Type: Single Cantilever
Temp. Range: 50 °C to 200 °C
β = 5 °C/min.
Frequency = 1Hz

DMA measures the modulus and damping (energy dissipation) properties of materials as the materials are deformed under a periodic stress or strain. These measurements provide quantitative and qualitative information about the performance of materials. DMA is particularly useful for evaluating mechanical properties of viscoelastic polymeric materials which exhibit time, frequency, and temperature effects.

PCB materials are comprised of fiberglass braid impregnated with a thermosetting resin. Characterization of the resin glass transition temperature can sometimes be difficult by DSC because of high fiber content. DMA, because of its inherent sensitivity to the glass transition, is an ideal technique for identifying the Tg of these highly filled systems. The plot below shows two sets of curves for a highly filled PCB material. The solid line curves represent the data for the material “as received”.The dashed line curves represent data for the “post baked” or annealed sample. Note the slight increase in Tg and the higher initial modulus value as a result of additional curing. This is a clear demonstration of DMA’s utility for characterizing the degree of cure and the final mechanical properties of filled PCB or thermosetting materials.

For more information or to place an order, go to http://www.tainstruments.com/ to locate your local sales office information

to view this article online, *click here*

BACK TO TABLE OF CONTENTS