

RF-10

with fastRise™

General Processing Guidelines

General Information

Ceramic/PTFE Laminate

RF-35TC, RF-35A2, RF-60TC, RF-10 and TSM-DS3 are thermally stable ceramic filled PTFE materials. They were developed to meet the demanding requirements for thermally stable dielectric constants and plated through-hole reliability in extreme environmental conditions.

AGC's ceramic/PTFE laminates are low loss, dimensionally stable, thin core materials for multilayer digital or RF applications that can be combined with our fastRise™ prepregs for the lowest stripline insertion losses. A lightweight style of fiberglass and very high loadings of ceramic particles yield excellent dimensional and electronic performance, as well as ease of processing. Most materials are available in thickness multiples of 0.005" (0.125mm), making it ideal for double-sided or multilayer applications.

Processing recommendations for ceramic/PTFE laminates are similar to other types of PTFE based AGC materials. It should be noted that these recommendations are based on standard industry practices and optimal parameters may differ somewhat, depending on available processing equipment.

fastRise™ Prepregs

Various laminate cores are used in conjunction with fastRise prepreg to produce multilayer boards for the RF/digital/ATE multilayer markets. fastRise, when used in a symmetrical board design, will result in optimum electrical and mechanical performance. Because of the thermoset properties of the bonding agent, multiple bonding cycles can be achieved without worry of delamination. In addition, the recommended press temperature of 420°F [215°C] is within reach of most board shops.

Storage

Ceramic/PTFE Laminate

Ceramic/PTFE laminates should be stored flat in a clean dry area at room temperature. Cores will benefit from being stored between two stiffeners in order to prevent unnecessary bending of the layers or damaged corners. Soft slip sheets should be used to prevent dust and debris from being embossed into the material. With proper storage conditions, cores have an indefinite shelf life.

fastRise™

Shelf Life

AGC will not ship fastRise™ material with less than 90 days of remaining shelf life. In the event that prepreg expires, please contact with your AGC technical sales person for assistance to coordinate retesting the expired prepreg.

Refrigeration

fastRise™ should be stored flat in a sealed bag below 42°F. Refrigeration temperatures of 14°F – 32°F are ideal for preventing unintended polymerization (cross-linking).

Upon removing fastRise™ prepreg from refrigeration, it should be allowed to acclimate to room temperature in a sealed bag. This will reduce the chance of moisture condensation on the prepreg and will also provide a more consistent start temperature for the lamination process. Bags should be resealed when not in use.

Handling

Ceramic/PTFE Laminate

PTFE is a thermoplastic material which is very stable electrically and chemically when compared with common thermosetting resins such as epoxy, polyphenyleneoxide, polyimide and cyanate ester. Part of what gives PTFE its superior performance over frequency and temperature also makes the pure resin relatively soft. It is for this reason that all AGC laminates are reinforced with glass fabric. The glass fabric reinforcement of the substrate greatly increases stability in the X and Y axis over non-woven or unreinforced PTFE products. While the glass fabric provides excellent dimensional stability, the following process and handling precautions should be taken to prevent damage or deformation of the laminate during fabrication.

Avoid mechanical scrubbing

As with thin core or flexible substrates, mechanically scrubbing will stretch and deform the material. The pinch rollers used to secure the panel during scrubbing will also cause dents as particle or brush material are pressed into the surface of the laminate. Chemical cleaning is much preferred. Eliminating mechanical cleaning and unnecessary handling will improve the dimensional accuracy of subsequent processes by preventing mechanical distortion of the laminate.

Do not pick up a panel horizontally by one end or edge

By allowing the material to flop over you may stretch the copper and substrate. Lift the panel by two parallel edges; preferably the two closest dimensionally.

Prevent contaminant deposits on the material or copper

The use of clean protective gloves and slip sheets will prevent contamination and staining. You will not need to remove oils, grease or fingerprints if you don't deposit them.

Do not mechanically abrade the PTFE surface after etching/removing the copper

If left undisturbed, the PTFE surface is very good for adhesion of solder mask, prepreg and bonding adhesive without further preparation. The etched surface of the PTFE is very wettable due to the rough tooth structure left behind after copper removal. If the surface becomes disturbed, further surface preparation using sodium or plasma etching can improve wettability and adhesion to the surface.

Do not stack panels directly on top of each other

Particles or debris on the surface of the panel can become imprinted into the copper and substrate of adjacent panels. The preferred method of storage is to rack the panels vertically. If panels must be stacked use clean, soft, slip sheet material between each panel and keep stack height to a minimum.

fastRise™ B-Stage

fastRise™ prepreg is supplied between two release sheets. The surface of fastRise™ may be tacky (especially for freshly manufactured material). Although it is recommended to allow refrigerated fastRise™ B-Stage prepreps to acclimate prior to opening a sealed bag, in some cases it may be advantageous to use the prepreg while it is cool* which will reduce the tackiness of the material and make handling easier.

*do not allow condensation to form on the prepreg

Inner Layer Preparation

Acclimation

If the laminate has seen extreme temperatures during shipment or storage, the material should be placed in ambient conditions for 8- 24 hours prior to processing.

Scaling

All laminates experience movement after the copper foil has been etched. Various factors such as laminate thickness, glass style, construction, copper foil thickness, and circuit design all contribute to the characteristic known as dimensional stability. The data is reference only and, again, is dependent on the factors listed above. Most printed circuit board shops determine artwork compensation data by running samples or estimating based on previous experience.

| Dimensional Change in Parts Per Million (PPM) | | |
|---|---------|---------|
| Material Designation | X | Y |
| TSM-DS | 200-400 | 200-400 |
| RF-35A2, 35TC | 200-400 | 200-400 |
| RF-60TC | 400-600 | 400-600 |
| RF-10 | 400-600 | 400-600 |

Lamination

Quick Start

The following chart is provided as a general starting point for lamination recipe development. The later sections provide substantially more detail allowing recipes to be optimized for specific applications.

| | Low Flow / Foil Lamination | Standard | High Flow / 30+ Layers |
|--|---|---------------|---------------------------|
| Vacuum | Full vacuum is recommended through entire cycle | | |
| Vacuum Delay | Hold vacuum 10-20 minutes before applying heat or pressure | | |
| Heating Rate | 3 - 8°F /min (1.7 – 4.4°C/min) | | |
| Critical Range | 176°F – 302°F (80°C – 150°C) | | |
| Pressure | 100 – 200 psi | 200 – 350 psi | 350 – 500 psi |
| Cure Temp / Time (measured at bondline) | 420°F -480°F for 60 – 120 minutes (215.5 – 248.9°C for 60 - 120 minutes) | | |
| Cooling Rate | Less than 6°F/min (3.3°C/min) | | |
| Breakdown | Breakdown or transfer to cold press when bondline is less than 200°F (93.3°C) | | |

*Please see the section on Foil Lamination which covers additional process parameters

Flow Patterns / Thieving

Solid copper borders with small alternating “star burst” vent lines are ideal. Interlocking thieving patterns, offset diamonds, honeycombs, or other patterns which inhibit resin flow channels are also acceptable. Interlocking “star burst” flow patterns or other patterns which may promote resin flow channel formation should be avoided.

Padding and Conformance Materials

Press padding (outside separator plates) is recommended. Use of conformance materials such as AGC TacPad, PTFE skive film, clutch lamination, or others are often helpful to balance pressure variations induced from circuits.

Pressure

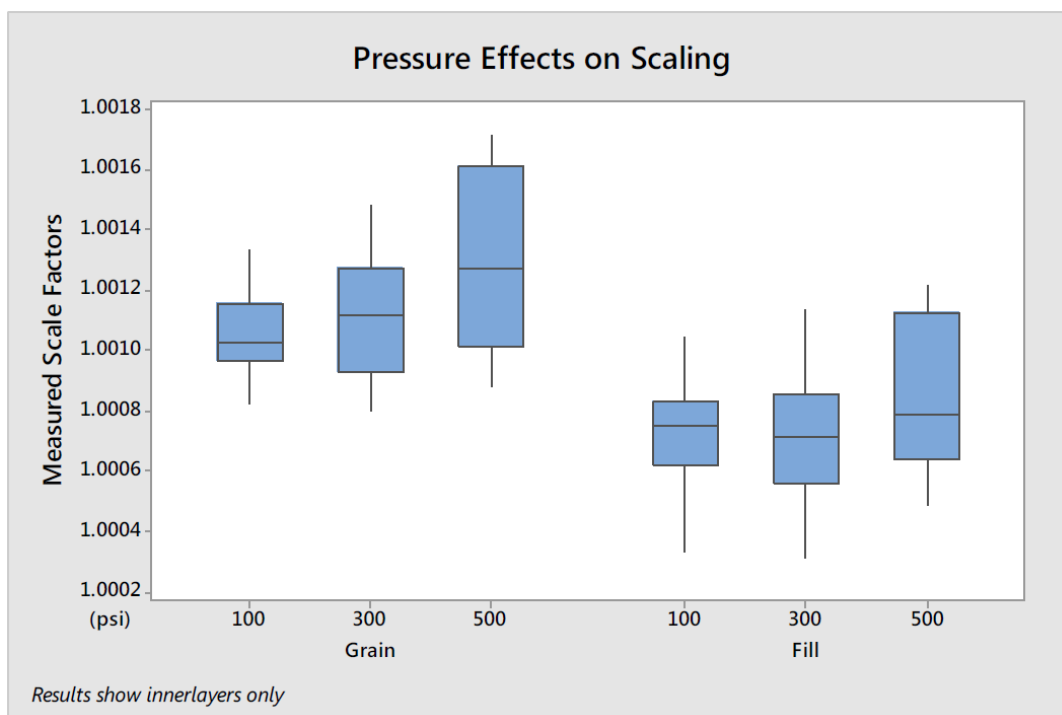
Excessive pressure should be avoided; it can distort circuit patterns, induce resin/filler separation, or create flow channels.

Full pressure should be achieved before the fastRise™ reaches 248°F (120°C). fastRise™ resin flow has been shown to be directly proportional to lamination pressure. Pressure can be used to control flow so that high-flowing fastRise™ part numbers will act as no-flows or low-flowing part numbers achieve additional flow.

For example, FR27-0035-26 resin flow can be approximated with the following equation:

$$\text{Flow \%} = \text{Pressure} \times 0.0002 \text{ [psi]}$$

Lower pressures have been linked to decreased grain-direction scale variation between panels (see Pressure Effects on Scaling below). This effect was not observed in the fill-direction. Although low pressures can be used to reduce grain-direction scale variation, it is advised to modify heating rates before reducing the pressures; heating rates have a larger impact on scale variation and the risk of inadequate pressure is avoided. Many fabricators find that a lamination pressure of 200-350 psi is effective for most products. In the event that high flow is required (e.g. heavy copper, via filling, etc.), some fabricators increase lamination pressure to 350-550 psi. Likewise, for low-flow applications (e.g. cavities, foil lamination, etc.) pressure ranges of 100-200 psi are often advantageous.



Low flow applications

Where limited or no resin flow is required, there are low resin flow formats of the fastRise™ and the engineer should consider using these types along with exploring how to manage the flow level with pressure and time in the low viscosity window, with the specific design of the PCB.

It is strongly recommended that suitable conformal press materials are used as well as employing methods for damming the resin flow in cut-outs or cavities.

Where cut-outs are made in the fastRise™, damming methods help with limited registration, restricted clearance to vias near the cavity, and gathering at the cut path when using mechanical processes. Although the minimum pressure should be used, pressure must be adequate to achieve adhesive contact surrounding the cutout area. Extending time in the low viscosity window may be helpful.

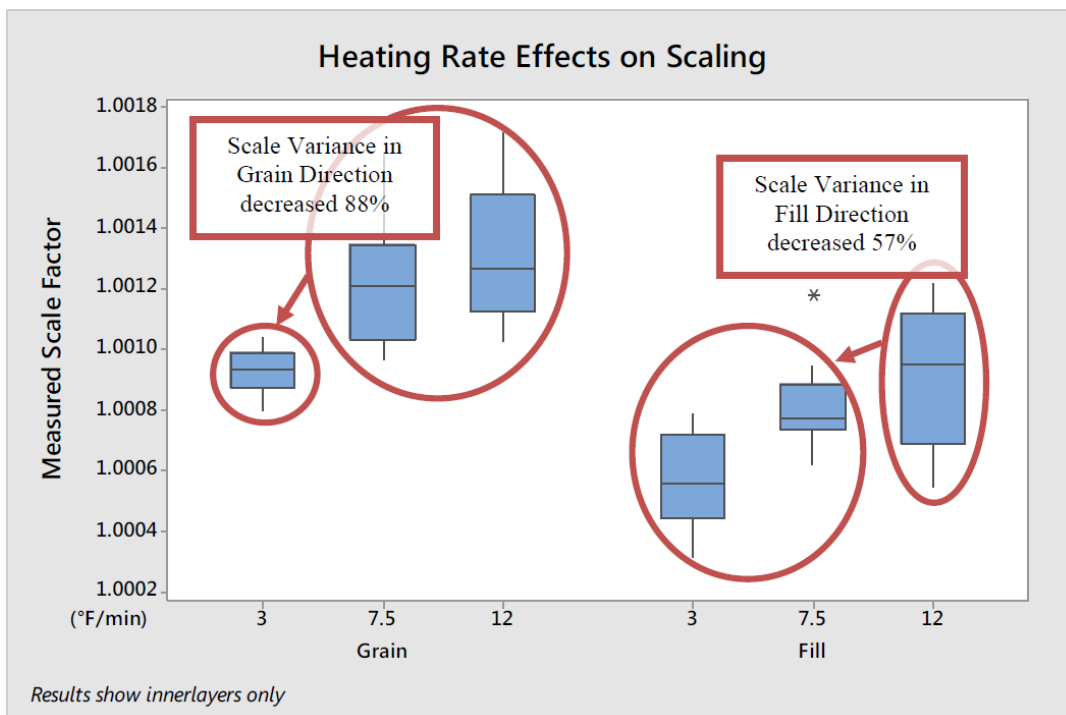
Temperature

Resin Flow Window / Critical Range

fastRise™ resin gels and melts between 176°F - 302°F (80°C – 150°C) and reaches its lowest viscosity between 215°F – 260°F (102°C – 127°C). Extending the time in which the resin is at its lowest viscosity can improve flow and is advantageous in hard to fill applications (e.g. heavy copper, embedded component cavities, via filling, etc.). Lengthening the time in this low viscosity window is accomplished by reducing the heating rate or by adding a dwell of up to 1 hour at 260°F (127°C).

Heating Rate

A cold start of the press is desirable. Typical fastRise™ heating rates are 3°F /min – 8°F /min (1.7°C/min - 4.4°C/min) and rates of 3°F /min – 10°F /min (1.6°C/min – 5.5°C/min) have been successful. In difficult to fill applications such as heavy copper (>1oz.), via filling, etc. or in high layer count boards, a slow heating rate should be used. It is strongly recommended that low heating rates be used if the process is to accommodate tight registration requirements or high layer counts. Data has shown that lower heating rates (i.e. 3°F/min) can provide substantial improvements in registration repeatability.



Curing

Curing begins at 395°F and fastRise™ curing processes are usually designed to hold the bondline between 420°F – 450°F for a minimum of 1 hour. Curing temperatures of up to 480°F can be used and the elevated temperatures will achieve performance gains while lower temperatures have also been successful in hybrid lamination cycles. AGC has observed that higher lamination temperatures combined with a 2 hour dwell can lower Zaxis CTE values by more than 30%. High-reliability applications often use a 2 hour cure at the elevated temperatures. The reduction in Z axis CTE is related to the resin content of the fastRise™ part number and will vary by fastRise™ part number. Elevated curing temperatures have also been shown to improve peel strengths. For this reason, it is strongly recommended that foil laminations use a cure temperature of at least 450°F. Hybrid laminations have successfully been accomplished by curing at 395°F for 3 hours. AGC advises caution as this is the minimum possible cure temperature and adhesion may be reduced.

Cooling

A slow cool is necessary to avoid any issues associated with delamination. The hot press should be cooled below 200°F before transferring to a cold press.

Foil Lamination

fastRise™ Selection

FR27-0040-43F and FR25-0021-45F were specifically designed for foil laminations and tend to provide superior surface quality and aesthetics. FR27-0040-43F is preferred due to the increased chances of drill smear with FR25-0021-45F. Other fastRise™ part numbers will also work with foil lamination, but additional process development may be required. Lower flowing part numbers typically yield better results.

Recommended Foils

TWS or TW foils from Circuit Foils is ideal. Other copper foils with sufficient dendrite tooth structure may also be acceptable.

Pressure

Foil lamination requires minimal pressure. 100-200 PSI should be adequate and higher pressures may be used to fill copper that is plated to 1-2 oz. If the surface appearance shows signs of excessive flow or flow channels, the pressure should be reduced. Pressure has a little to no impact on peel strength. Cure temperature is much more effective at improving bond strengths.

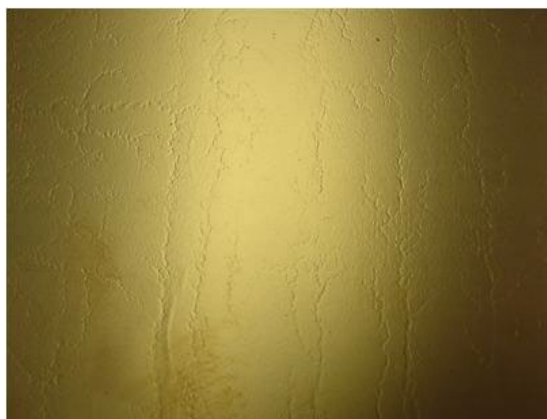
Temperature

Slower heating rates are recommended for foil laminations and an added 60 minute dwell at 225°F (107°C) may also be advantageous to allow the resin to flow uniformly without flow channels. When heating from 225° (107°C) to 450°F (232°C), much lower heating rates of 3°F/min – 5°F/min should be used. A cure temperature of 450°F (232°C) for a minimum of 1 hour should be used for foil laminations. Peel strengths increase with elevated cure temperatures and extended cure times. Cooling rates should not exceed 6°F/min (3°C/min) until bondline is less than 200°F (93°C).

Foil Lamination Surface Quality

Excessive resin flow can cause flow channels which may be visible on the foil surface after lamination. If the adhesion is good in some areas but delamination is found in other areas, it is likely the result of thermal shear stresses and there are several steps which can be taken. Cooling should never be done by transferring the laminates from a hot press to a cold press. Cooling rates of 2°F/min – 4°F/min or lower are ideal. An aluminum sheet and/or conformal padding (e.g. AGC TacPad, skive PTFE, PacoPlus, etc.) placed between the panel and the steel plate will help reduce transferred shear stress. Press padding, such as PacoPad will also help.

Several fabricators have found clutch laminations helpful in which a layer of higher flowing prepreg (e.g. FR4, acrylic, polyimide) is placed between release sheets (e.g. PTFE film, foil release, PVDF film) that are then placed against the part during lamination. The higher flowing prepreg will flow providing hydrostatic pressure as well as offsetting thermal stresses.



Additional Notes

Poor Pressure Distribution

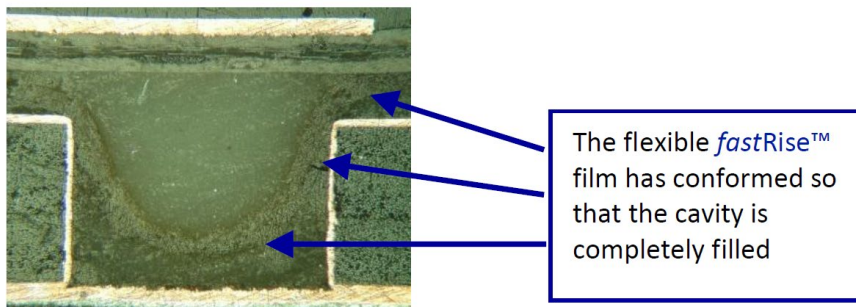
Constructions containing very low pressure areas with stacked regions of little or no copper between layers (e.g. coupons, ground clearances, fiducials, etc.) may require special considerations to achieve ideal bonding. Resin from the higher flowing fastRise™ part numbers combined with higher pressures will fill the low pressure areas to balance the pressure. Although the low flow fastRise™ may be capable, caution is advised to ensure that adequate resin flow and resin volume are present to avoid poor adhesion and resin starvation. Copper distribution should be as balanced as possible especially with thicker or plated circuits.

Multiple Ply Constructions

Resin flow can increase dramatically if multiple plies of fastRise™ are used against each other. If high-flow is not desired, pressure should be reduced to approximately 25%-50% to prevent excess resin flow, resin separation, or formation of flow channels.

Encapsulating the conductor pattern

Although fastRise™ film is not porous, it is flexible and will conform to circuit patterns. The film can have an elongation of 30-300% and will fill circuit patterns as long as there is sufficient resin to flow within the areas.



Drilling

Quick Start

The following chart is provided as a general conservative starting point for drill process development. The later sections provide substantially more detail allowing the process to be optimized for differing applications.

Please contact Technical Service for a Drill Table Creator file to calculate specific process parameters.

| | Imperial units | SI units |
|--------------------------------------|--|---|
| Entry Material | Aluminum or Phenolic (0.010" – 0.015" preferred) | Aluminum or Phenolic (0.25mm - 0.38mm preferred) |
| Backer Material | Rigid Phenolic, Slickback, or comparable | |
| Cutting Speed (Surface Speed) | 100 SFM | 30.5 MPM |
| Chip Load | 0.0010 in | 25.4 μm |
| Dwell | 0-1000 ms (increase dwell time as speed and chip load deviate from above recommendations) | |
| Hit Count | 50-300 | |

Drill Bits

Standard 130° point geometry, 32° - 45° helix angle PCB carbide drills work well with all AGC ceramic/PTFE-based laminates. Stack height should not exceed 2/3 the flute length of the smallest diameter drill being used.

Sharp drill bits are critical to any PTFE drilling; new drill bits should always be used. Undercut drill bits are recommended, but past studies have shown that some drill bit brands may obtain better results using their standard drill bits.

| Vendor | Model | Helix Angle | Style | Nodules/hole |
|---------------|-----------------------|-------------|----------|--------------|
| Union | NEU L026, 0.25x4.5 | 40 | UC | 0.54 |
| Union | MD35 A3815, 0.25x5/#8 | 35 | Straight | 0.72 |
| Kemmer | SH381010021021R | 38 | Straight | 0.67 |
| Kemmer | E34000250-C45040 | 38 | UC | 0.72 |
| Kyocera/Tycom | 460.0098.177 | 38 | UC | 0.98 |
| Kyocera/Tycom | 560.0098.177 | 38 | Straight | 0.12 |
| HPTec | 8 212 0250 | 40 | UC | 0.48 |
| HPTec | 212, 157mil flute | 45 | UC | 0.25 |

Chip Load

A nominal chip load of 1.0 mil (25.4 µm) is a recommended starting point for all tool sizes. Increasing or decreasing the chip load from 1.0 mil (25.4 µm) has not been found to offer any quality improvements. Increasing the chip load to 1.25 mils (31.8 µm) may provide acceptable hole quality and improved productivity. If drill smear is visible and bits are sharp, reduce chip load until it is eliminated.

Cutting Speed

Drill speeds of 100 SFM (30.5m/min) or less is recommended. Slower speeds offer the greatest hole-quality improvements; they allow generated heat to dissipate before smearing PTFE. Drill speed can be increased due to equipment limitations, but added dwell times may become more important. If drill smear is visible and bits are sharp, reduce cutting speed until it is eliminated.

Dwell Time

Lower surface speeds will reduce or eliminate the need for dwell. If ideal cutting speeds cannot be obtained, a 250ms dwell is recommended for initial process setup in order to cool the drill bit between holes and prevent softening of the PTFE that will later smear across interconnects. Past AGC studies have shown that hole-wall quality may improve as dwell times are increased to as much as 1000ms.

Peck Drilling

Peck drilling should be avoided where possible; it has been shown to increase drill bit wear as well as increase process time. Peck drilling may be required in some situations (e.g. bird nesting, hole plugging, chip extraction on thick panels, breaking thin drill bits, etc.). Peck drilling with a full withdrawal of the drill bit after each peck will reduce heat buildup and debris accrual. A general rule of thumb for peck depth is 20 to 30 mils per peck and should be optimized at the board shop. If traditional peck drilling is not used, hole-wall quality may be improved with the use of a "clean" peck where the peck depth is set to equal that of the Phenolic entry. In this, the entry material will effectively clean the drill bit, retract to clear Phenolic debris and cool, and then reenter to drill the hole.

Past studies have shown that peck drilling ceramic/PTFE based PCBs is necessary for micro diameters (12mil and 14mil) to remove enough debris in order to prevent tool breakage. However, for 28mil holes, it is not clear that peck drilling <0.200" ceramic/PTFE based PCBs is required. Some drill studies suggest that peck drilling will leave a small circular ring where the drill bits stop in the hole.

Hit Count

AGC recommends using new drills for the best hole quality. Hole wall quality will be directly related to drill bit hit count. AGC's ceramic/PTFE laminates have very high loadings of ceramic particles. A conservative hit count is 150 hits/bit. 200-300 hits/bit can be used for less critical application. 50-100 hits/bit might be necessary if drill smear is a problem.

Due to the ceramics used in TSM-DS3, RF-35TC, and RF-60TC, close attention must be paid to drill bit wear. Minimize hit counts as necessary to maintain hole wall quality.

For very demanding hole wall specifications it might be necessary to drill the holes, pass the PCB through an electroless treatment or flash plating treatment to essentially make the debris in the hole rigid, then re-drill to snap the debris from the side of the hole wall. Another strategy for critical hole wall quality is to slightly under drill (using undersized diameter drill bit) the hole, thereby removing most of the PTFE-ceramic from the hole, flash plate the hole, and then re-drill it with the proper size bit. If undersized drilling is used, flash plating between drill sizes may not be necessary.

Entry / Backer Materials

Standard Phenolic (≥ 0.020 " thickness) or aluminum (0.007"-0.015") entry material is acceptable. A hard Phenolic backup board (from 0.060" to 0.125" thickness) is recommended to reduce bottom-side burring.

Aluminum or Phenolic entry serves to abrade Teflon debris off the bit. Optimizing the amount of Phenolic to clean the drill tool between hits prevents bird nesting and smear, and is a necessary factor in high quality ceramic/PTFE drilled holes. Do not worry about Phenolic causing excessive drill wear. Although the process window is wider for drilling double sided PCBs vs. multilayer PCBs, one should follow the same strategies and tactics that one would use to fabricate a multilayer based on ceramic/PTFE cores.

The pressure of the drill foot should be a minimum of 40 psi and should be increased if topside burring is excessive.

Laser Drilling / MicroVias

fastRise™

The lasing of the fastRise™ part numbers will vary between the different part numbers; please consult the fastRise™ datasheet for details regarding the ideal part numbers. Standard fastRise™ part numbers typically yield improved hole-wall quality.

FR-28-0050-40

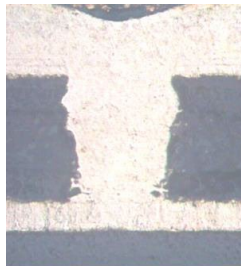


Photo by Hughes Circuits

Hole Wall Preparation

fastRise™ and ceramic/PTFE laminates require a PTFE activation cycle. If AGC laminates are combined with fastRise™, panels will benefit from a desmear/etchback process performed prior to the PTFE activation.

Desmear

Plasma

If panels have been exposed to moisture, bake the boards at 220°F - 250°F (104°C - 121°C) for 1 hour. Standard FR4/epoxy desmear processes should then be used. The desmear plasma time is typically half that of standard FR4/epoxy times because fastRise™ resin system tends to etch back very quickly.

Permanganate

Permanganate desmear IS NOT RECOMMENDED and has been shown to be very aggressive on fastRise™ resulting in excessive etchback. This is due to the high silica filler content and thermoset content in the resin system.

PTFE Activation

Plasma

If panels have been exposed to moisture, pre-bake the boards at 220°F - 250°F (104°C - 121°C) for 1 hour. Plasma treat the PTFE resin using 70%/30% Hydrogen/Nitrogen gas mixture. 100% Helium may also suffice. Power settings for the RF-signal generator are typically 60-75% of full rated power for 30-60 minutes. Thick panels or high-aspect ratio holes may require extended plasma cycle times. Thick panels may also benefit from an additional 30 minute O2 plasma process prior to the PTFE activation plasma. Experience has shown that gases such as Helium and CF4 are not as effective as Hydrogen as evidenced by sporadic plating voids and higher contact angles. The advantage of plasma etching is that it is a relatively safe procedure. Disadvantages include relatively long cycle times (35 to 60 minutes) and short shelf life of the effect (4 – 24 hours).

Sodium Etch

Sodium Etches (e.g. Fluoroetch) work well with both fastRise™ and AGC's ceramic/PTFE laminates. Follow the manufacturer's recommended treatment process. Subsequently, bake for 1 hour at 250°F[121°C] prior to plating to remove moisture that may have been absorbed during the sodium treatment process. The advantages of sodium etching include long shelf life of the hole wall treatment, fast treatment time, and complete coverage. The primary disadvantage of sodium etchant is the volatility of the chemical.

Chlorine can have adverse effects on the sodium treatment. Do not subject exposed sodium etch treated holes to heavily concentrated chlorine-based chemical processes.

Process Example

The following table is offered by March Plasma as a basic starting point recipe:

| Power (kW) | Pressure (mT) | Gases | Gas Ratios | Flow (slm) | Pl Temp (°C) | Time (minutes) | Function |
|------------|---------------|----------------------------------|------------|------------|--------------|----------------|---|
| 4.5 | 250 | O ₂ / N ₂ | 90 / 10 | 2.5 | 90 | A/R | Heating |
| 4 | 250 | CF ₄ / O ₂ | 10 / 90 | 2.5 | 99 | 10 | Thermoset etch-back |
| 4 | 250 | O ₂ | 100 | 2.5 | 99 | 5 | Removes fluorine and cleans the glass |
| 4.2 | 250 | N ₂ | 100 | 2.5 | 99 | 30 | Activates PTFE. H ₂ /N ₂ Cycle is typically more effective and reliable |

Note: Regardless of which method of hole wall treatment is used, desmearing of the thermoset resin should be done prior to treatment of the PTFE resin.

Plating

After the hole wall has been properly prepared, fastRise and the ceramic/PTFE laminates will accept either electroless copper or direct metallization plating. The electrolytic plating process is the same for PTFE or epoxy based materials. Typical plating consists of 1 – 1.5 mils [25µm - 35µm] of copper plate in the holes and/or on the surface. For high-aspect ratios or other difficult to plate applications, a second pass through the electroless process may be required to ensure proper hole-wall coverage. It may also be beneficial to run a short duration of electrolyzed copper, rinse etc., then restart the electrolyzed copper from the beginning to expose the hole wall to fresh chemistry.

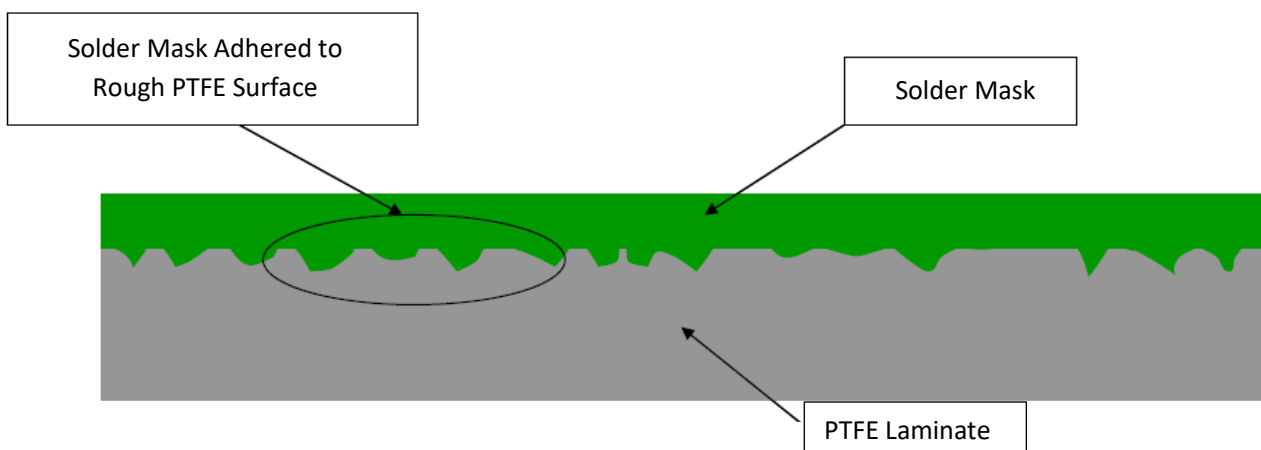
Image, Develop, Etch, Strip

Prepare the copper surface, apply dry film, and image and develop using a standard process. The copper surface preparation should consist of microetching the copper. Mechanical scrubbing (e.g. pumice scrub) is not recommended for thin core PTFE-based materials or multilayer inner layers due to possible registration issues and mechanical damage. The etching process is the same as for a standard printed circuit board. Machine settings should be appropriate for the copper thickness of the multilayer inner layers. Strip the photoresist using a standard process.

Solder Mask

Overview

With the advantages of liquid photo imageable (LPI) soldermasks being their ease of use and resolution capability, they often do not have the same adhesion characteristics as the previous pure epoxy systems. This fact must be taken into account when applying LPI's to PTFE laminates. The mechanism for adhesion of soldermask (or prepreg or bonding film) to the PTFE laminate surface is the condition of the PTFE surface prior to application. By nature, PTFE is a very low surface energy fluoropolymer and thus it has excellent non-stick properties, which make it highly popular in lubrication and release applications. However, the adhesion of the base copper cladding is achieved by lamination of the relatively rough treated copper surface to the PTFE material under high heat and pressure. This process is able to produce an excellent mechanical bond between the PTFE resin and the rough dendritic surface of the copper. It is the negative impression of the rough copper treatment that remains in the PTFE after etching the copper to form the circuitry pattern, which provides adequate surface area for mechanical bonding of the soldermask to the PTFE surface. Therefore it is important to eliminate traditional scrubbing techniques which may disturb or destroy this rough surface.



With the need to replace the process a substitute process has been used which has several added benefits. Chemical cleaning of the copper surface offers the advantage of reduced mechanical stress on the material. This becomes critically important when dealing with thin laminates (<0.010" or 25 mm) and or critical dimensional tolerances in soldermask or second stage drilling or routing. The removal of the scrubbing application also eliminates pits and dents which may be caused by high pressure contact with the steel or ceramic coated steel pinch rollers typically used in scrubbing machines to prevent panel movement during rotary scrubbing. In rare cases where adhesion is poor, a plasma treatment may be used to activate the exposed PTFE surface.

Pattern Plated Copper

1. Etch panels as normal to define circuitry pattern.
2. Allow plated metal etch resist (tin or tin/lead, in the case of nickel or gold surfaces obviously these will remain) to remain on panel through post etch inspection processes to prevent copper surface contamination, staining or oxidation prior to soldermask application.
3. Set up soldermask application process, prior to stripping or removing the metallic etch resist. Note: For double sided soldermask application, set up solder mask process to apply soldermask to the side of the panel with the largest copper area to be covered (i.e. if the ground plane is to be completely covered with soldermask then this would be the first side coated).
4. Strip tin or tin/lead from copper surfaces. The copper surfaces should be bright and stain free following tin or tin/lead strip. Note: Omit this process for nickel and gold finishes.
5. Chemically clean and roughen the copper surfaces using an acid or alkaline cleaner followed by a micro-etch process which removes 30-60m" of copper. This should provide adequate surface area for adhesion of soldermask to copper surfaces.
6. Dry panels thoroughly. If you do not have an adequate horizontal dryer, then an oven bake is recommended at 150-170°F [66-77°C] for 15-20 minutes.
Note: Drying process should not cause oxidation of copper surfaces. If oxidation occurs reduce drying time. PTFE Laminate Solder Mask Solder Mask Adhered to Rough PTFE Surface PTFE Surface.
7. Allow panels to cool to room temperature (approximately 5-10 minutes) and immediately apply soldermask.
8. Tack dry (LPI) or bake and cure (silk screened epoxy) soldermask per manufacturer's recommendation.
9. Continue processing per manufacturer's recommendation for LPI soldermasks (image, develop and cure).
10. For second side, repeat processes beginning with # 5.

Panel Plated Copper

1. Etch panels as normal to define circuitry pattern.
2. Strip dry film resist from panels and rinse and dry thoroughly to prevent copper surface oxidation.
3. Using clean white cotton gloves, perform post etch inspection immediately following dry film removal. Move panels to soldermask process directly after inspection.
4. Set up soldermask application process while post etch inspection is taking place to minimize hold time between etch, strip and soldermask application.
Note: For double sided soldermask applications, setup soldermask process to apply soldermask to the side of the panel with the largest copper area to be covered (i.e. if the ground plane is to be completely covered with soldermask then this would be the first side coated).
5. Chemically clean and roughen the copper surfaces using an acid or alkaline cleaner followed by a micro-etch process which removes 30-60m" of copper. This should provide adequate surface area for adhesion of soldermask to copper surfaces.
6. Dry panels thoroughly. If you do not have an adequate horizontal dryer, an oven bake is recommended at 150-170°F [66-77°C] for 15-20 minutes.
Note: Drying process should not cause oxidation of copper surfaces. If oxidation occurs reduce drying time.
7. Allow panels to cool to room temperature (approximately 5-10 minutes) and immediately apply soldermask.
8. Tack dry (LPI) or bake and cure (silk screened epoxy) soldermask per manufacturer's recommendation.
9. Continue processing per manufacturer's recommendation for LPI soldermasks (image, develop and cure).
10. For second side, repeat processes beginning with step # 5.

Solder Replow

Hot air solder leveling (HASL) is a common method of protecting exposed copper circuitry. Two basic types of hot air level machines are used in the industry, horizontal and vertical. By far the more popular of the two is the vertical machine. It is less expensive and easier to maintain than the horizontal type. However, the vertical machine subjects the printed circuit board to a more severe level of thermal shock than the horizontal.

Once the board is clamped in the vertical machine, the only preheat the board sees is the few seconds above the solder pot prior to immersion. Typical immersion times are 5 – 6 seconds from entry into the solder pot to complete withdrawal. This minimal preheat time can be particularly harsh on PTFE based laminates due to their z axis expansion characteristics. When using hot air solder leveling on PTFE laminates, AGC recommends a bake cycle of 2 – 3 hours at 300°F[150°C] just prior to the HASL process.

The solder pot temperature should be maintained at 460°F - 480°F[238°C – 250°C] for optimal performance. Cycle time should be 5 – 6 seconds from the time of entry to the complete withdrawal of the board. Dwell time in the solder pot should not exceed 2 seconds. Longer preheat times and adjusted cycle times may be advantageous depending on design and processes.

ROUTING / MILLING

Machining of ceramic/PTFE substrates is typically more difficult than epoxy-based substrates due to the softness of the PTFE resin system. The style of fiberglass used in the substrate also affects the quality of routing with respect to burrs and fibers. The heavier the fiberglass weave, the more difficult it is to cut. AGC's ceramic/PTFE laminates can be successfully machined using two flute end mills when the recommended methods and rout parameters are used. Rigid Phenolic entry and a rigid backer should be used.

In some cases, adding paper (white paper or Kraft paper) between the Phenolic and the part allows better conformance to surface topography (e.g. circuits, soldermask, etc.) and may reduce burring. For tight tolerances or superior edge quality, a "rough cut" placed 0.005 in. -0.010 in. off the part edge may be run prior to the "finish" cut at the nominal part edge. Historical data have yielded the following recommended rout parameters.

These parameters are good starting points for all standard AGC materials. Special materials such as those with heavy metal ground planes may require different rout parameters and are not addressed in this guideline.

| Router Diameter | | Spindle Speed | Feed Rate | |
|-----------------|------|---------------|-----------|---------|
| (mils) | (mm) | (kRPM) | (in/min) | (m/min) |
| 31.5 | 0.8 | 50 | 11.8 | 0.25 |
| 35.4 | 0.9 | 45 | 11.8 | 0.27 |
| 39.4 | 1.0 | 40 | 11.8 | 0.32 |
| 43.3 | 1.1 | 37 | 11.8 | 0.33 |
| 47.2 | 1.2 | 34 | 11.8 | 0.34 |
| 51.2 | 1.3 | 31 | 15.8 | 0.37 |
| 55.1 | 1.4 | 29 | 15.8 | 0.41 |
| 59.1 | 1.5 | 27 | 15.8 | 0.43 |
| 63.0 | 1.6 | 25 | 19.7 | 0.45 |
| 66.9 | 1.7 | 24 | 19.7 | 0.53 |
| 31.5 | 0.8 | 50 | 11.8 | 0.25 |
| 35.4 | 0.9 | 45 | 11.8 | 0.27 |
| 39.4 | 1.0 | 40 | 11.8 | 0.32 |
| 43.3 | 1.1 | 37 | 11.8 | 0.33 |
| 47.2 | 1.2 | 34 | 11.8 | 0.34 |
| 51.2 | 1.3 | 31 | 15.8 | 0.37 |
| 55.1 | 1.4 | 29 | 15.8 | 0.41 |
| 59.1 | 1.5 | 27 | 15.8 | 0.43 |
| 63.0 | 1.6 | 25 | 19.7 | 0.45 |
| 66.9 | 1.7 | 24 | 19.7 | 0.53 |

These guidelines can provide only basic and reference information for PCB fabricators. Because of different environment, equipment, tooling and so on, in all instances, the user shall determine suitability in any given conditions or applications. For more detailed processing information, please contact with the AGC engineer or sales representative.