

RF-60TC

with fastRise™

General Processing Guidelines

General Information

High Thermal Conductivity Laminates

RF-35TC and RF-60TC are highly thermally conductive, ceramic-filled, glass-reinforced PTFE laminates.

Due to their exceptional loss characteristics, they are best suited for high power applications where every 1/10th of a dB is critical and the PWB substrate is expected to diffuse heat away from both transmission lines and surface mount components such as transistors or capacitors. Because they are PTFE based, they also exhibit a stable dielectric constant across a wide range of operating temperatures. RF-35TC and RF-60TC 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 performance. Both materials are available in thickness multiples of 0.005" (0.125mm), making them ideal for double-sided or multilayer applications.

Processing recommendations for RF-35TC and RF-60TC laminates are roughly the same for other types of ceramic-filled/PTFE materials with a few noted exceptions. These recommendations are based on standard industry practices. Optimal parameters may differ based 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 lamination 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

RF-35TC / RF-60TC Laminate

RF-35TC and RF-60TC 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™

Store the material in a cool dry area away from direct sunlight and high humidity, avoiding material contamination.

fastRise™ is certified to meet all requirements as agreed upon between the user and supplier for a given shelf life as defined by the storage conditions below.

Storage Conditions

Condition 1 (i.e. refrigeration): <4.5°C (40°F)

Condition 2 (i.e. room temp): <23°C (73°F), Relative Humidity <50%

When removing fastRise™ prepreg from refrigeration, it should be allowed to acclimate to room temperature in the 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.

Shelf Life

If material is stored under Condition 1 above, a shelf life of 180 days after receipt of shipment will apply. If material is stored under Condition 2 above, a shelf life of 90 days after receipt of shipment will apply. AGC will not ship fastRise™ material with less than 90 days of remaining shelf life. Packaging will default to indicate shelf life based on storage Condition 2 unless we are notified that Condition 1 applies. In the event that prepreg expires, please contact your AGC technical sales person for assistance to coordinate re-testing the expired prepreg.

Handling

RF-35TC / RF-60TC 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 most 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 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 will allow for good adhesion of solder mask, prepreg, photoresists, and other bonding adhesives 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™

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™ prepregs 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, 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	Machine Direction	Cross Direction
RF-35TC	400-600	200-400
RF-60TC	400-600	200-400

Lamination

Quick Start

The following chart is provided as a general starting point for lamination recipe development using RF-35TC or RF-60TC laminates with fastRise™ prepregs. 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 (2 - 4°C/min)	3 - 8°F/min (2 - 4°C/min)	2 - 4°F/min (1 - 2°C/min)
Critical Range	175°F – 300°F (80°C – 150°C)		
Pressure	100 – 200 psi	200 – 350 psi	350 – 500 psi
Cure Temp / Time (measured at bondline)	450°F (230°C) for 60 – 120 minutes	420°F (215°C) for 60 minutes	420°F (215°C) for 60 minutes
Cooling Rate	Less than 6°F/min (3°C/min)		
Breakdown	Breakdown or transfer to cold press when bondline is less than 200°F (90°C)		

Flow Patterns / Thieving

Solid copper borders with interlocking thieving patterns such as offset diamonds, honeycombs, or other patterns which inhibit resin flow channels are ideal. Interlocking “star burst” flow patterns or other patterns which may promote resin 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 250°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. 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 350-500 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 flow versions of fastRise™ (part numbers ending in “F”) that the design engineer should consider using. If they are unable to accommodate these part numbers in their design, the fabricator may explore how to manage the flow level of standard fastRise™ part numbers with lower pressures and less time in the low viscosity window. 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 175°F - 300°F (80°C – 150°C) and reaches its lowest viscosity between 215°F – 260°F (100°C – 125°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 (125°C).

Heating Rate

A cold start of the press is desirable. Typical fastRise™ heating rates are 3°F/min – 8°F/min (2°C/min – 4°C/min) and rates of 3°F/min – 10°F/min (2°C/min – 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 (200°C). fastRise™ curing processes are usually designed to hold the bondline at or above 420°F (215°C) for a minimum of 1 hour. Cure temperatures of up to 450°F (230°C) can be used, while lower temperatures of 395°F (200°C) have also been successful in hybrid lamination cycles. AGC has observed that higher lamination temperatures combined with a 2 hour dwell can lower Z-axis 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 marginally improve peel strengths. For this reason, it is strongly recommended that foil laminations use a cure temperature of at least 450°F (230°C). Hybrid laminations have successfully been accomplished by curing at 395°F (200°C) 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 (90°C) before transferring to a cold press.

Foil Lamination

fastRise™ Selection

FR-27-0040-43F and FR-25-0021-45F were specifically designed for foil laminations and tend to provide superior surface quality and aesthetics. FR-27-0040-43F is preferred due to the increased chances of drill smear with FR-25-0021-45F. Other fastRise™ part numbers will also work with foil lamination; however 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. The low flow part numbers can withstand normal pressure ranges up to 400 psi without developing flow channels. 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 260°F (125°C) may also be advantageous to allow the resin to flow uniformly without flow channels. When heating from 225°F (110°C) to 450°F (230°C), much lower heating rates of 3°F/min – 5°F/min should be used. A cure temperature of 450°F (230°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 (90°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. FR-4, 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

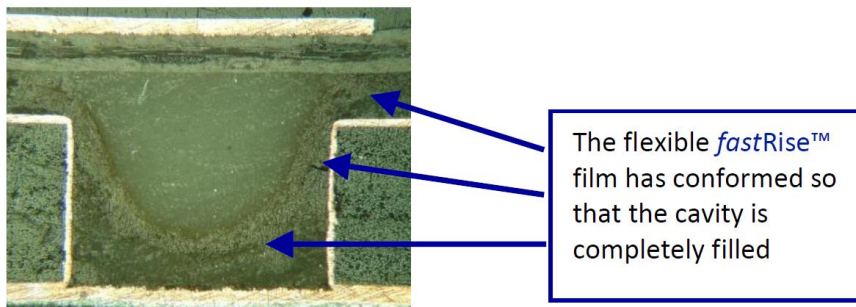
Constructions containing very low pressure areas with stacked regions of little or no copper between layers (e.g. coupons, ground clearances, fiducials, etc.) may require special considerations to achieve ideal bonding. Resin from the higher flowing fastRise™ part numbers combined with higher pressures 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 by 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 and Phenolic (0.024" preferred)	Aluminum and Phenolic (0.60mm 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	25-100	

Drill Bits

Standard 130°-point geometry, 32° - 45° helix angle PCB carbide drills are sufficient to drill RF-35TC and RF-60TC laminates. Due to the highly abrasive nature of the ceramic mix used to achieve high thermal conductivity, it may be economical to consider the use of diamond-coated carbide drills for critical vias drilled in these materials. Contact your AGC technical service representative for further guidance. 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.

Chip Load

A nominal chip load of 0.001" (25 µm) is a recommended starting point for all tool sizes. Some fabricators have found success in the range of up to 0.002" (50 µm). This will vary depending on types of 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. machines and drill bits being used and should be tested or proven out at each facility. Non-critical vias (RF ground stitching or heat dissipation vias) will obviously have wider processing windows. Higher chip loads generally produce more heat on the drill bit, which will lead to drill smear if not properly addressed.

If drill smear is visible and bits are sharp, reduce chip load until it is eliminated.

Cutting Speed

Drill speeds of 100 SFM (30 m/min) are recommended. Slower speeds offer the greatest hole-quality improvements; they allow generated heat to dissipate before smearing PTFE. In certain stack ups, drill speed can be increased to 150-200 SFM (45-60 m/min) to improve productivity without sacrificing quality 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.

If tool breakage is an issue for small diameter, high aspect ratio holes, peck drilling may be inevitable. However, for 0.020" holes with an aspect ratio of roughly 12:1 or less, it is not clear that peck drilling 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. To obtain high thermal conductivity while remaining economical to produce, RF-35TC and RF-60TC employ the use of a particularly abrasive ceramic filler. A conservative hit count with normal carbide bits is 50 hits/bit. 100-200 hits/bit can be used for less critical application. Less than hits/bit might be necessary if drill smear is a problem. Hit counts may improve in hybrid stack ups with only 1 or 2 thin cores of the TC material. Paying close attention to drill bit wear in these scenarios may provide longer drill life without sacrificing quality. 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 (0.024" thickness) and aluminum (0.007"-0.015") entry material is recommended. Thicker phenolic (0.048" +) may be necessary on panels thicker than 0.200". A phenolic backup board (from 0.040" to 0.125" thickness) such as LCOA Spectrum Gold is recommended to reduce bottom-side burring and drill smear. It is critical to fully plunge the drill bit into the phenolic backup to clean off debris before retracting the bit through the PCB and potentially re-depositing it on the hole wall. The addition of lubricated backups, such as Slickback, are optional but do provide some marginal effectiveness at reducing heat buildup on the drill bit, which ultimately leads to drill smear. Aluminum and phenolic serve 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 RF-35TC and RF-60TC 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 RF-35TC and RF-60TC 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

RF-35TC and RF-60TC Laminates

RF-35TC and RF-60TC are ideally laser drilled with a dual source laser (UV/CO2). It is possible to use a UV laser only at the potential expense of hole wall quality and reliability. Because of the high ceramic loading, most of the material removal is completed with the UV laser and the CO2 laser serves to remove any potential remaining PTFE from the microvia hole walls and the capture pad. The high ceramic loading also limits the manufacturability of microvias to a 0.005" thick laminate. Creating a microvia in a 0.010" thick laminate will require either a 2-step laser program (to dissipate heat), or a mechanical controlled depth drill (to remove the bulk of the material) followed by a laser drill to complete the microvia.

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 and those ending in "S" typically yield improved hole-wall quality. Shown below is a laser via through the FR-28-0040-50 part number.

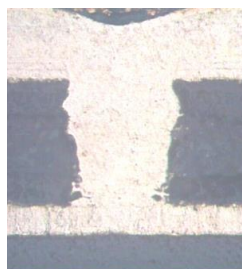


Photo by Hughes Circuits

Hole Wall Preparation

RF-35TC and RF-60TC laminates require a PTFE activation cycle. If they are combined with fastRise™ in a multilayer PCB, the panels will benefit from a desmear/etchback process performed prior to the PTFE activation as directed below.

Desmear

Plasma

If panels have been exposed to moisture, bake the boards at 220°F - 250°F (105°C - 120°C) for 1 hour. Standard FR-4 desmear processes (CF₄/O₂) should then be used. The CF₄ desmear plasma time is typically half that of standard FR-4 times because the fastRise™ resin system tends to etch back very quickly.

Permanganate

A permanganate desmear IS NOT RECOMMENDED if the process contains glass etch chemistry. This is due to the high ceramic content of the fastRise™ resin system and will result in excessive etch back. Standard permanganate and glass etch baths as a part of the electroless copper process are OK. However, note that this alone will not sufficiently desmear the fastRise™ resin. A plasma process as described above is required for good hole wall quality.

PTFE Activation

Plasma

If panels have been exposed to moisture, pre-bake the boards at 220°F - 250°F (105°C - 120°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 RF-35TC / RF-60TC laminates. Follow the manufacturer's recommended treatment process. Subsequently, bake for 1 hour at 250°F (120°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)	Pnl 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	H ₂ / N ₂	70 / 30	2.5	99	30	Activates PTFE. Alternative is 100% He gas

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, TSM-DS3 with fastRise™ 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.

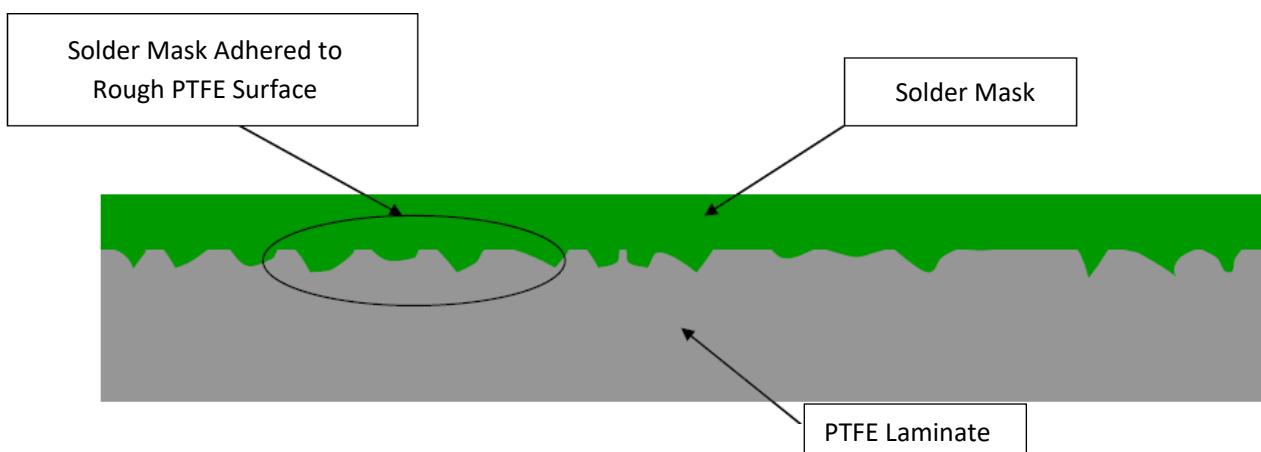
Image, Develop, Etch, Strip

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 TSM-DS3 as the abrasion of the etched PTFE surface removes the left over surface topography required for good adhesion to photoresists. It may also cause distortion and registration issues if performed on inner layers or thin core subassemblies. 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 0.25 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) 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.
5. Chemically clean and roughen the copper surfaces using an acid or alkaline cleaner followed by a micro-etch process which removes 30-60 μin 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 [65-75°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 just enough so that they may be handled (approximately 5 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 30-60 μin 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 (65-75°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 just enough so that they may be handled (approximately 5 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 Reflow

When using hot air solder leveling on RF-35TC or RF-60TC, 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° – 480°F (240°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 RF-35TC and RF-60TC is typically more difficult than epoxy-based substrates due to the softness of the PTFE resin system combined with the abrasiveness of the ceramic filler. The low fiberglass content of these laminates does provide the advantage of reducing the amount of burring or exposed fibers left on the board edge. RF-35TC and RF-60TC 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 PCBs comprised primarily of RF-35TC or RF-60TC. Special materials such as those with heavy metal ground planes may require different rout parameters and are not addressed in this guideline. More frequent bit changes are recommended. Router bit life should be established during initial setup of the routing process.

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
70.8	1.8	23	23.6	0.60
74.8	1.9	21	23.6	0.63
78.7	2.0	20	27.6	0.68
82.7	2.1	20	31.5	0.76
86.6	2.2	20	31.5	0.80
90.6	2.3	20	31.5	0.84
94.5	2.4	20	35.4	0.88
98.4	2.5	20	35.4	0.92
118.1	3.0	20	43.3	1.06
125	3.18	20	43.3	1.10

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.