AGC
Your Dreams, Our Challenge

RF-35TC

General Processing Guidelines

General Information

RF-35TC, RF-35A2, and TSM-DS3 are Thermally Stable ceramic filled Materials. The TSM-DS3 is developed to meet the demanding requirements for dielectric constant temperature stability and plated hole reliability for high temperature and varying environmental conditions. TSM-DS3 are low loss, dimensionally stable thin core material for multilayer digital applications, that can be combined with our fastRise prepregs for the lowest stripline insertion losses. The TSM-DS3 uses a light weight style of fiberglass and very high loadings of ceramic particles yielding excellent dimensional and electronic performance, as well as ease of processing. TSM-DS3 are available in thickness multiples of 0.005" (0.125mm), making it ideal for double-sided or multilayer applications.

Processing recommendation for the TSM-DS3 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.

Handling

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.

Do not mechanically scrub the material
As with thin core or flexible substrates, mechanically scrubbing will stretch and deform the material. The pinch rollers used to move the panel during scrubbing can 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 cooper 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.

Drilling for Double Sided Ceramic Filled PTFE Laminates

Good hole quality can be achieved using AGC recommended drill parameters. Standard 130° point geometry, 32 - 35 helix angle PCB carbide drills work well with all AGC PTFE-based laminates. Hole quality can be affected by drill sharpness. AGC recommends using new drills for the best hole quality. Hole wall quality will be directly related to drill bit hit count. TSM-DS3 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 applications…50-100 hits/bit might be necessary if drill smear is a problem. Stack height should not exceed 2/3 the flute length of the smallest diameter drill being used. Standard phenolic entry material (≥0.020" [0.5mm] thickness) is acceptable. A hard phenolic backup board (from 0.090" to 0.125" [2.29mm – 3.18mm] thickness) is recommended to reduce bottom-side burring. The pressure of the drill foot should be a minimum of 40 psi and should be increased if topside burring is excessive. The primary objective of drilling PTFE based laminates is avoiding bird nesting. Debris on the drill bit runs the risk of smearing off onto the hole and adhering to the hole as the drilled hole and drill tool become hot and the PTFE acts like an adhesive. Hard phenolic on entry serves to abrade Teflon debris off the bit. Optimizing the amount of phenolic to clean the drill tool between hits is a necessary factor in high quality TSM-DS3 drilled holes. Although the process window is wider for drilling TSM-DS3 double sided pwbs vs multilayer pwbs, one should follow the same strategies and tactics that one would use to fabricate a multilayer based on TSM-DS3 cores.

AGC recommends the following drill conditions for double sided ceramic filled PTFE material:

- Entry material: Aluminum sheet or standard phenolic entry (10-15 mil Phenolic preferred).
- Backup material: Hard phenolic of thickness from 0.060" to 0.125".
- Stack height should not exceed 2/3 the flute length of the smallest diameter drill being used.
- Drill parameters as found in the drill appendix. For microdrilling (<0.020" diameter) use the highest speed column parameters that equipment allows.
- Optimize drill parameters to reduce or eliminate birdnesting, burrs, and smearing.
- Hit counts range from 50 hits to 500 hits depending on drill size and board thickness.
- Peck drilling if necessary.

All drilling debris must be removed prior to hole wall preparation such as plasma or sodium treatment. Thoroughly remove all debris in the holes with a high pressure air or water blast. If water is used, bake the laminate for 1 hour at 250°F [120°C] to remove moisture prior to through hole treatment. Burring can occur if drilling conditions are not correct. If burring occurs, sanding is not recommended. Pumice scrubbing has been known to be effective, however AGC does not recommend any process that may cause distortion of the laminate. The best solution to prevent burrs is by thoroughly understanding and implementing the optimum drilling process and parameters for your equipment. Smearing is a condition where the PTFE resin has been heated to a point where it softens and is easily moved within the hole. It usually appears as a line between the copper foil and the plated copper. Assuming that sharp drill bits are being used, the solution is to reduce the cutting speed (thus surface feet per minute) and infeed rate to prevent the heating up of the drill bit. Low chip loads are the key to success in drilling PTFE. If debris builds up on the bits (birdnesting) it is necessary to optimize the thickness of the phenolic entry material. Phenolic entry material is a hard material that will abrade PTFE off the surface of the bit. If the drill tool is contaminated with PTFE in the flutes, the drill debris will strike an inner layer and smear drill bit debris across the post. The phenolic entry material should be used as a sacrificial material to smear drill bit debris onto the phenolic so that the drill bit reaches the PTFE laminate core free of any debris. Dwell times between hits will allow the bit to cool before reentry. Protrusions may not be noticeable until after the electroless plating process, even if a high magnification microscope is used. See Section 6 for hole-wall preparation. Hole wall tear-out, or gouging, is another possible defect caused during the drilling process. Gouging is usually an indicator of either a dull drill or an excessively high chip load. Another factor that can influence gouging is the fiberglass weave style. A coarse glass is more prone to gouging than a medium or fine glass style.

If gouging occurs, first check for worn drill bits before adjusting the drill parameters. If gouging persists, reduce the chip load. Care should be taken to keep drill bit temperatures to a minimum by keeping the cutting speed low while adjusting chip loads downward. Due to the thickness of high layer count digital multilayer boards, peck drilling may be required for acceptable hole wall quality. The thermoplastic nature of the PTFE resin and the thickness of the board may result in the inability of the drill bit to clear debris which may result in smearing and burrs. 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.

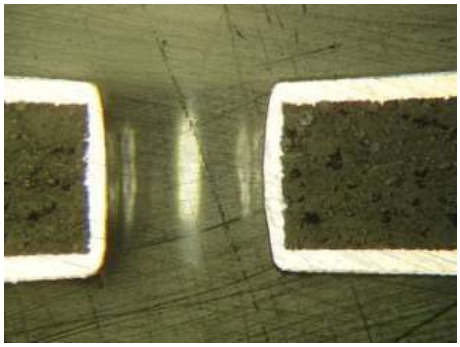


Figure 1 – Cross section illustrates the very low fiberglass content of TSM-DS3

Drilling TSM-DS3 and Multilayers Containing fastRise Prepreg

The typical material combination for RF / Microwave and digital multilayers are TSM-DS3 and fastRise27 and one of the fastRise27 part numbers. Tight process control and optimized parameters are essential to achieve required hole wall quality for thick multilayer (> 0.100") and back plane. High speed digital application has tight specification to ensure that press fit connectors are not damaged.

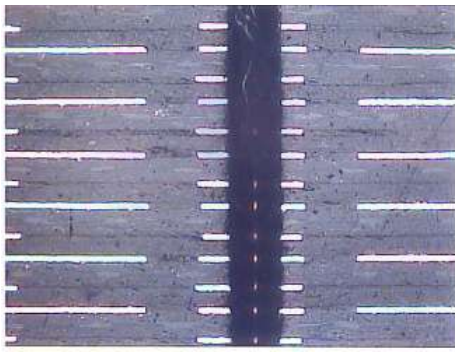


Figure 2 - Multilayer containing TSM-DS3 and fastRise27

Drill setup conditions

(1) Drill bits: Use only new straight shank bits. For example: a. Diameter < 15mil Tycom/Kyocera 560 series straight shank. A comparison of drill bit vendors yielded the following results for a 10 mil tool when drilling a 115 mil TSM-DS3/FR27 multilayer stackup:

Vendor	Model	Helix Angle	Style	Nodules/hole
Union	NEU L026, 0.25 x 4.5	40	UC	0.54
Union	MD35 A3815, 0.25 x 5 / #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

Parameters: 10 mil Tycom 560 series straight shank, 7 mil aluminum entry, 16.5 mil phenolic entry, 16.5 mil phenolic exit, 120K speed, infeed 55, 0.25 second dwell between hits, 1st peck to the depth of phenolic + aluminum + 2 mils into material (28 mil), 1 peck/30 mils thereafter.

(2) Maximum hit counts: 200 hits/bit

(3) Drill Stackup Illustration (for 250 mil thick pwbs, thick phenolic entry should be used):

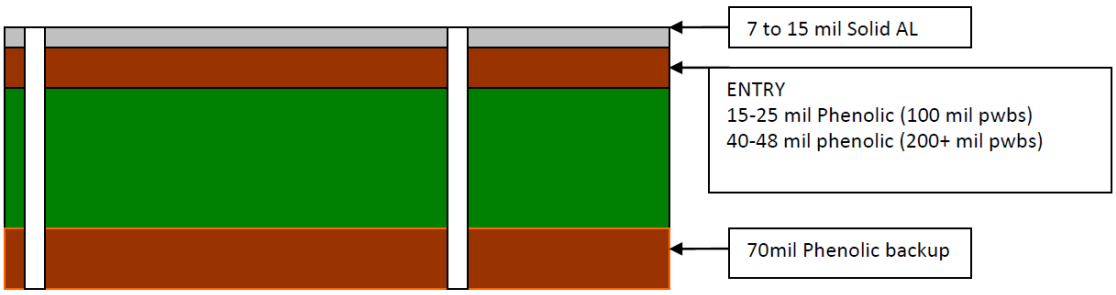


Figure 3 – Stackup of Multilayer Drilling

(4) Vacuum: Maximum capacity
(5) Drill foot pressure: > 40 psi
(6) Dwell between hits: 1 second (250 millisecond may work)
(7) Peck drilling through the phenolic entry material.

Table 1. TSM-DS3 Double Sided or Multilayer Drill parameters with fastRise.

Drill Size (inch)	Chipload (mil)	Spindle Speed (Krpm)	Feed Rate (ipm)	Retract Rate (in/min)
0.0100	0.4 – 0.6	80 – 125	55 – 60	1000
0.0120	0.4 – 0.6	80 – 125	55 – 60	1000
0.0200[1]	0.64	70	45	1000
0.0200[2]	0.33	45	15	1000
0.0280	1.0	32	32	1000
0.0380	1.3	24	31	1000
0.0400	1.8	22	40	1000
0.0420	1.8	21	38	1000
0.0440	1.8	20	37	1000
0.0460	1.8	19	35	1000
0.0480	1.8	19	34	1000
0.0500	2.0	19	36	1000

[1] 50 hits per bit with peck drilling
[2] 150 hits per bit, no pecking

Entry/Exit Materials

Phenolic entry and exit material is critical for drilling PTFE based circuit boards. 16-25 mil phenolic is recommended for 100 mil pwbs. 40-50 mil phenolic entry is recommended for 250 mil thick pwbs. PTFE gets hot and likes to accumulate on the surfaces of the bit. Phenolic entry and exit material will essentially “clean” the surface of a bit. Debris builds up on the bit which includes anything the bit comes into contact with: entry material, copper, fiberglass, silica, PTFE, metal from the bit itself, phenolic, Aluminum etc. It is necessary to use a hard thick phenolic entry material to essentially break the debris off the bit and smear the debris onto the phenolic before the bit arrives clean at the surface of the pcb and drills. Following these drill parameters should allow the user to arrive at fairly good hole wall quality on very thick telecom type boards (200-300 mil).

DO NOT WORRY ABOUT PHENOLIC CAUSING EXCESSIVE DRILL WEAR. Drilling Teflon is limited by debris filling and fusing to the drill bits. Drill bits are generally exchanged due to debris build up on the bits before wear becomes an issue. Reducing hit count should eliminate wear as a factor. Typically 70 mil phenolic is used as backup but the phenolic only needs to be 15 mils or the depth of the drill. If the drill only penetrates 20 mils at the bottom the phenolic need only be 20 mil. Phenolic backup can be stacked with spectrumboard. Phenolic can also be combined. A 25 mil piece of phenolic can be combined with a 15 mil piece of phenolic to yield a 40 mil piece of phenolic. If the requirements are such that even higher hole wall quality is needed there is another solution:

Tip: Strategies to remove plating nodules: 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 redrill 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 remove most of the PTFE-ceramic from the hole, flash plate the hole, and then redrill it with the proper size bit. If undersized drilling is used, flash plating between drill sizes may not be necessary.

Dwell Times

Studies have shown that the best hole wall quality is obtained when the drill bit has a chance to cool. This makes intuitive sense because a cooler bit will be less likely to cause the PTFE to become soft and stretchy. We have found that a 1 second (1,000ms) dwell time between hits is sufficient. Longer dwell times will not improve hole wall quality.

Peck Drilling

We have also found that it is beneficial to "clean" the bit by pecking to a depth that corresponds to the thickness of the entry material. Pecking through the distance of the phenolic will abrade debris off the bit in the phenolic and allow the debris from the phenolic to be removed before the bit reenters the hole and starts drilling the multilayer based on fastRise. Therefore the first peck should be the depth of the entry material. This is necessary to remove the phenolic debris before "cleaned" drill bit starts drilling into the PTFE based pcb. Past studies have shown that peck drilling inside of PTFE pcb is necessary for micro diameters (12mil and 14mil) to remove the debris in order to prevent tool breakage. However, for 28mil holes, peck drilling inside 200 mil PTFE based pcb is still not clear. Some drill studies suggest that peck drilling will leave a small circular ring where the drill bits stop in the hole. However, it may be necessary to peck drill to remove drill debris. Peck drilling has the benefit of allowing the bit to cool a bit and it improves debris removal. However, we have found in a very complex 250 mil pwb that the peck drilling puts a lot more wear on the drill bit. You need a sharp cutting tool to cut the Teflon smoothly otherwise it does smear. Some drill bit vendors claim that pecks increase drill wear proportional to the # of pecks. No pecking and low 50 hits per bit has the best chance of eliminating smear because the drill bit will stay very sharp. To drill a thick pwb with no pecking however, you have to operate at very low chiploads. Table 4 shows two different successful drill parameters for drilling a 20 mil hole, with and without peck drilling.

Inner Layer Preparation

Multilayer applications require that two or more laminates be bonded together to form a single circuit board. There are two important considerations when processing the circuit board. The first is registration of the features from one layer to another. The second is the condition of the bond surface prior to lamination.

Registration: Layer to layer registration is often a critical requirement of the finished circuit board and misregistration can cause a variety of issues such as open circuits and poor coupler performance. Therefore, it is important that the material be acclimated to the processing environment and the correct artwork compensation used. Acclimation to the processing environment is simply making sure that the laminate is at ambient temperature prior to processing. It is recommended that if the laminate has seen extreme temperatures during shipment or storage, it should be placed in ambient conditions for 24 hours prior to processing. 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.

Material Designation	Dimensional Change in Parts Per Million (PPM)	Material Designation	Dimensional Change in Parts Per Million (PPM)
TSM-DS3	200 - 400	TLY	400 – 800
TLC	200 – 400	**RF-35, 35P**	**200 – 400**
TLE	220 – 400	RF-60	400 – 600
TLT	400 – 600	CER-10	400 – 600

Table 2 – Typical dimensional change

Hole Wall Preparation

PTFE based laminates require that drilled holes be subjected to a process which will prepare the PTFE resin system for subsequent plating. There are two processes that have been shown capable of providing void-free copper plating. One process is chemical in nature and involves a sodium-based solution that strips the fluorine atoms from the PTFE molecules. This process has been used for many years in the industry with great success. 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. Various manufacturers and etchant services are available. Another method of preparing the holes for plating is plasma etching. If the proper gases and cycles are used, plasma will allow plating to the PTFE resin. Experience has shown that the best gases to use are a mixture of hydrogen and nitrogen. Helium can also be used in many cases. 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 Treatment: A sodium-based chemical treatment process does an excellent job of preparing the PTFE through-hole surface prior to the plated through hole process. Follow the manufacturer's recommended treatment process. 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.

NOTE: Do not subject the treated holes to heavily concentrated chlorine-based chemical processes prior to electroless copper plating or direct metallization. Chlorine can have adverse effects on the sodium treatment and result in plating voids.

Plasma Etching: Plasma treat using a 30-70% Hydrogen, 70-30% Nitrogen gas mixture. Power setting for the RF-signal generator should be 60-75% of full rated power for 30-60 minutes depending on the hole diameter, number of holes, and thickness of the board. Boards with relatively higher aspect ratios will require longer plasma cycle times. Industry 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.

Special treatment should be used if TSM-DS3 are combined with AGC's fastRise. fastRise also contains a thermoset resin as the bonding agent. This requires a desmear process in addition to the PTFE treatment. There are two common methods of desmearing thermoset resin systems. One of the desmear processes involves dipping the boards in a permanganate chemical which dissolves the resin. The second method is plasma. Generally speaking, the gases used to desmear the thermoset resin do not affect the PTFE resin, nor do the gases used for treating the PTFE resin desmear the thermoset resin very well if at all. Also, the sodium solution commonly used to treat the PTFE resin will not affect the thermoset resins system. Because TSM-DS3 contain no thermosetting resin…..they must be treated with either plasma gases or sodium etching for proper wet out and adhesion of the plating chemicals.

For multilayer applications, Permanganate desmear IS NOT RECOMMENDED and has been shown to be very aggressive, due to the relatively thin layer of thermosetting resin and due to the fact that the thermosetting layer contains a silica filler, and can cause excessive etchback. Permanganate will not affect the PTFE. Desmearing of the fastRise can be more closely controlled with a plasma desmear cycle. In addition, hole wall treatment of the PTFE can also be done successfully with plasma and it is possible to have back-to-back desmear and hole treatment without removing the product from the plasma machine. Basically desmear and PTFE activation should occur in a two cycle plasma cycle. TSM-DS3 core materials will not be affected by a permanganate desmear process.

AGC recommends the following hole wall treatment:

- Thoroughly clean holes prior to treatment. If water cleaning is used, bake the boards at 220°F (105°C) for 1 hour
- Plasma treat the epoxy component using a standard epoxy desmear gases and power and a 15 minute cycle time to reduce overetching of the epoxy
- Continue plasma cycle with a PTFE cycle shown below

Plasma Etching: If desmearing or etchback of the thermoset used to bond fastRise is necessary, the prepreg should first be exposed to a standard FR4 plasma desmear (10-15 minutes, 185°F /85°C, O2, N2 CF4 gas mixture. If too much etchback or pencil tipping of the interconnect occurs at the post then the dwell time in the FR4 plasma desmear cycle needs to be reduced. The treatment of the PTFE requires a different gas mixture. Plasma treatment of the PTFE resin using a 30-70% Hydrogen, 70-30% Nitrogen gas mixture has been shown to be very effective. If Hydrogen is not available, a 100% Helium should also suffice. Power setting for the RF-signal generator should be 60-75% of full rated power for 30-60 minutes depending on the hole diameter, number of holes, and thickness of the board. Boards with relatively higher aspect ratios will require longer plasma cycle times. Industry 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.

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 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.

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. Scrubbing is not recommended for thin core PTFE-based materials or multilayer inner layers due to possible registration issues.

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

Soldermasking of PTFE materials can be achieved quite easily once the processes of cleaning, application and the adhesion mechanisms are thoroughly understood. The following covers these fundamentals. Soldermask materials have changed drastically in the last 10 years with a move from two component fixed pot life epoxies, to single component heat cured epoxies, to the now standard liquid photo imageable soldermasks. The LPI's are different in that they incorporate a UV reactive component which allows these modified acrylic formulas (or combinations of epoxy and acrylates) to be imaged photographically for fine line resolution between fine pitch pads.

With the advantages of LPI's 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 (Table 3) 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 (Figure 3). It is the negative impression of the rough copper treatment that remains in the PTFE after etching the copper to form the circuitry pattern (Figure 4), which provides adequate surface area for mechanical bonding of the soldermask to the PTFE surface (Figure 5). Therefore it is important to eliminate traditional scrubbing techniques which may disturb or destroy this rough surface.

Copper Type	RzDin (microns) "Peak"	Ra (microns) "Average"
1⁄4 oz. ED copper	4.95	0.80
1⁄2 oz. ED copper	6.65	1.00
1 oz. ED copper	9.60	1.50
2 oz. ED copper	9.70	1.55

Table 3 – Typical copper treatment roughness for various standard weights of electrodeposited copper foils.

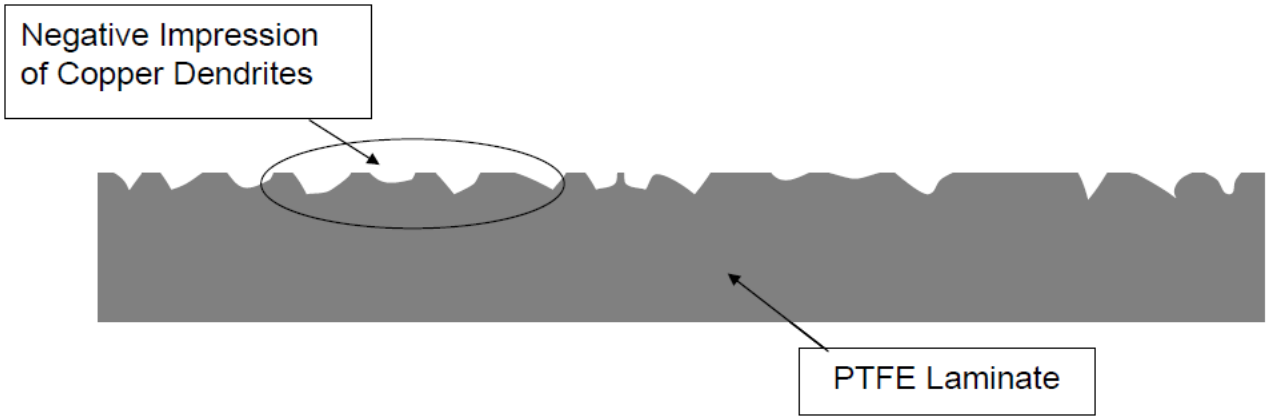


Figure 4 – Illustration of PTFE surface remaining after copper removal / etching.

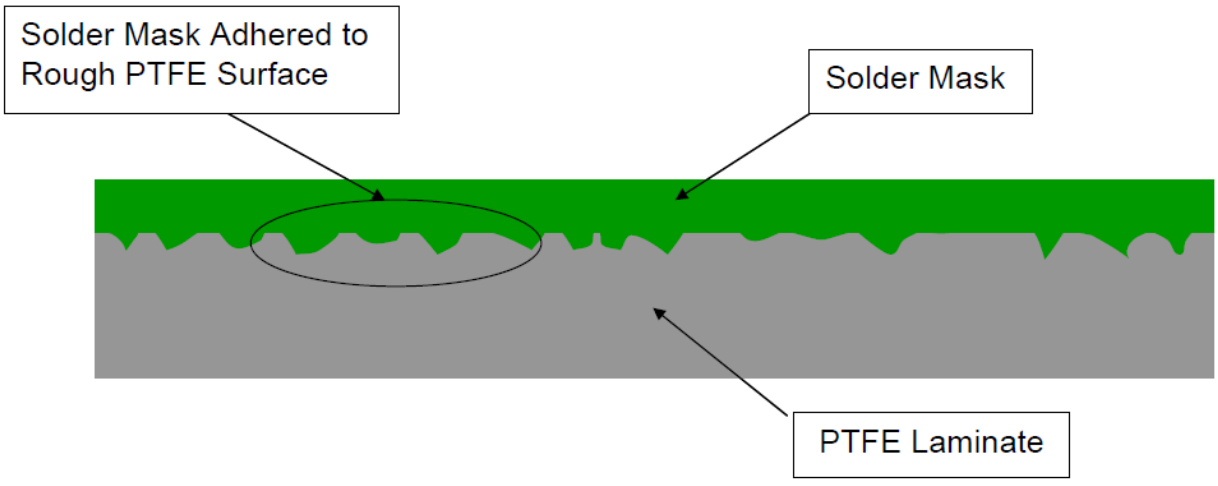


Figure 5 – Illustration of soldermask wetting and adhesion to rough PTFE surface.

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.

Soldermask Process – 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 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).

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 30-60µ" 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 drier 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.

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.

Soldermask Process – 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 30-60µ" 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 drier 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.

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 Reflow

Hot air solder leveling 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 (149°C) just prior to the HASL process. The solder pot temperature should be maintained at 460° - 480°F (238° - 249°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.

Machining / Routing

Machining of PTFE-based 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 products can be successfully machined using two flute end mills when the recommended methods and rout parameters are used. In addition to the rout parameters, an equally important factor in successful routing is having intimate contact throughout the routed package. Figure 6 shows a typical rout stack with phenolic entry and backer material on either side of the circuit board. Notice that the entry material rides on top of the copper traces leaving an air gap between the entry material and the PTFE substrate. In this case, the router bit will force the soft substrate into the air gap at the area circled in red.

Many circuit board applications also have soldermask on top of the copper traces which increases the gap further. The solution to a cleaner cut is to introduce a material between the copper traces and the phenolic entry material that will conform at the edge and will help fill in the air gap. One type of paper that has been shown to work well is the paper found in between artwork film. It is thick enough to fill in the normal air gap and cuts without generating as much debris as other paper such as Kraft paper.

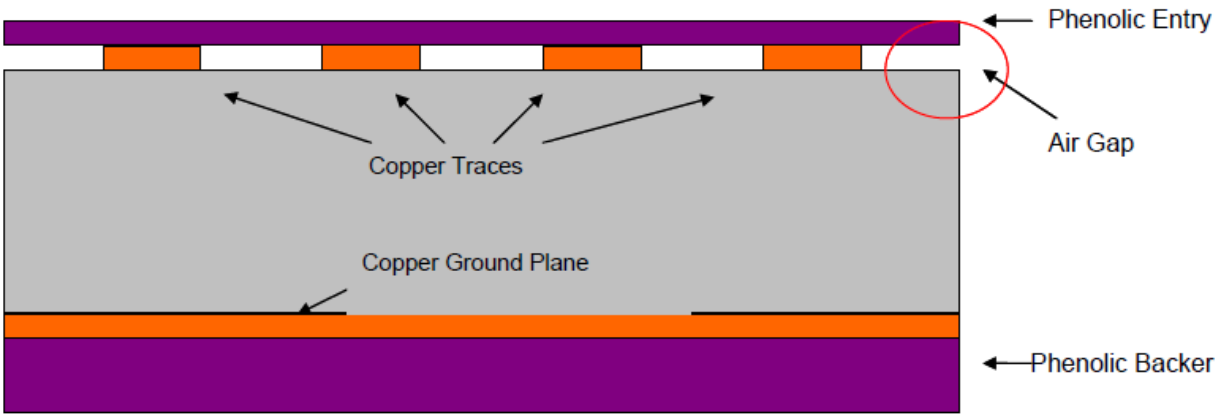


Figure 6 – A typical rout stack with phenolic entry and backer material on either side of the circuit board

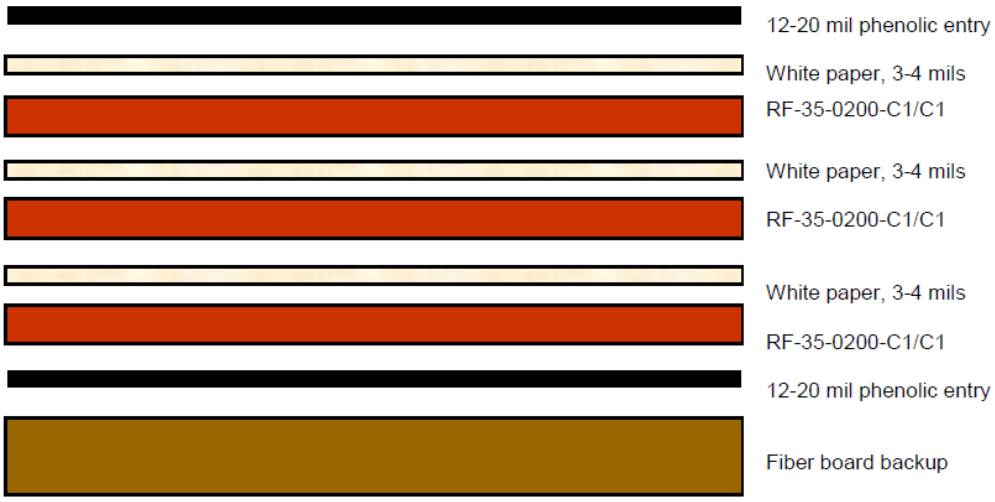


Figure 7 – Typical TSM-DS3 rout stack-up

Recent routing trials using AGC materials 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.

Recommended Routing Parameters for All AGC Materials							
Tool Diameter (mils)	Chipload (mils/rev)	Spindle Type				Z - Feed Rate	
		60,000 max		80,000 max		Without predrilling (in/min)	With predrilling (in/min)
		Spindle Speed(rpm)	Feed Rate (in/min)	Spindle Speed(rpm)	Feed Rate (in/min)		
31.5	0.24	50,000	11.8	50,000	11.8	0.0	20
35.4	0.26	45,000	11.8	45,000	11.8	0.0	20
39.4	0.30	40,000	11.8	40,000	11.8	0.0	79
43.3	0.32	37,000	11.8	37,000	11.8	0.0	79
47.2	0.35	34,000	11.8	34,000	11.8	0.0	79
51.2	0.51	31,000	15.8	31,000	15.8	0.0	79
55.1	0.54	29,000	15.8	29,000	15.8	0.0	79
59.1	0.59	27,000	15.8	27,000	15.8	0.0	79
63.0	0.79	25,000	19.7	25,000	19.7	0.0	197
66.9	0.82	24,000	19.7	24,000	19.7	0.0	197
70.8	1.03	23,000	23.6	23,000	23.6	0.0	197
74.8	1.12	21,000	23.6	21,000	23.6	0.0	197
78.7	1.38	20,000	27.6	20,000	27.6	0.0	197
82.7	1.58	20,000	31.5	20,000	31.5	0.0	197
86.6	1.58	20,000	31.5	20,000	31.5	0.0	197
90.6	1.58	20,000	31.5	20,000	31.5	0.0	197
94.5	1.77	20,000	35.4	20,000	35.4	0.0	197
98.4	1.77	20,000	35.4	20,000	35.4	0.0	197
118.1	2.17	20,000	43.3	20,000	43.3	0.0	197
125.0	2.17	20,000	43.3	20,000	43.3	0.0	197

Table 4 – Recommended routing parameters for all AGC materials

Recommended Routing Parameters for All AGC Materials							
Tool Diameter (mm)	Chipload (µm/rev)	Spindle Type				Z - Feed Rate	
		60,000 max		80,000 max		Without predrilling (m/min)	With predrilling (m/min)
		Spindle Speed(rpm)	Feed Rate (m/min)	Spindle Speed(rpm)	Feed Rate (m/min)		
0.80	5	50,000	0.25	50,000	0.25	0	0.50
0.90	6	45,000	0.27	45,000	0.27	0	0.50
1.00	8	40,000	0.32	40,000	0.32	0	2.00
1.10	9	37,000	0.33	37,000	0.33	0	2.00
1.20	10	34,000	0.34	34,000	0.34	0	2.00
1.30	12	31,000	0.37	31,000	0.37	0	2.00
1.40	14	29,000	0.41	29,000	0.41	0	2.00
1.50	16	27,000	0.43	27,000	0.43	0	2.00
1.60	18	25,000	0.45	25,000	0.45	0	5.00
1.70	22	24,000	0.53	24,000	0.53	0	5.00
1.80	26	23,000	0.60	23,000	0.60	0	5.00
1.90	30	21,000	0.63	21,000	0.63	0	5.00
2.00	34	20,000	0.68	20,000	0.68	0	5.00
2.10	38	20,000	0.76	20,000	0.76	0	5.00
2.20	40	20,000	0.80	20,000	0.80	0	5.00
2.30	42	20,000	0.84	20,000	0.84	0	5.00
2.40	44	20,000	0.88	20,000	0.88	0	5.00
2.50	46	20,000	0.92	20,000	0.92	0	5.00
3.00	53	20,000	1.06	20,000	1.06	0	5.00
3.18	55	20,000	1.10	20,000	1.10	0	5.00

Table 5 – Recommended routing parameters for all AGC materials (metric)

Multilayer

General Information for TSM-DS3 multilayers with fastRise Prepreg

fastRise prepreg is part of the AGC product offering designed specifically for High Speed Digital /ATE (DUT) applications, and RF multilayers for mmWave frequencies. Coupled with other AGC laminate cores, fastRise enables the use of low electrical loss stipline structures in high multilayer count boards out to high data rates (43+ gbps) and mmWave frequencies.

The thermoset properties of the bonding agent enable multiple laminations and thermal excursions during standard circuit board fabrication without delamination of the multilayer package. fastRise utilizes a high loading of ceramic for low electrical loss and dimensional stability, a high performance thermoset resin as a bonding agent for the copper foil, and some PTFE. The major component is silica. fastRise prepreg can be used to bond together all standard PTFE core materials (TLY, RT/Duroid 5880 etc), ceramic filled PTFE material (TSM-DS3, RO3003, RT/duroid6002, CLTE), rubber products (RO4003/R4350), standard modified epoxy cores (N4000-13SI), and polyimide cores.

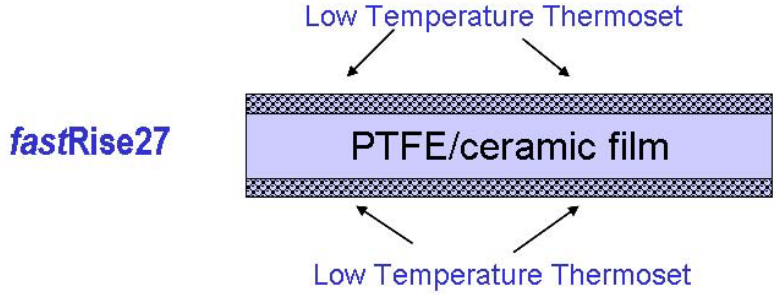


Figure 8 – fastRise construction

fastRise is designed very much like an acrylic based polyimide prepreg. Polyimide prepregs typically contain a polyimide film with a butter coat of a low temperature adhesive. fastRise contains a highly ceramic loaded PTFE film with a surface coat of a low temperature adhesive. The adhesive layer is responsible for flowing and filling between the innerlayer artwork. There are two mechanisms for filling the artwork. The thermosetting resin that is coated onto the PTFE/ceramic film will do most of the work in filling the copper artwork as shown in the pictures below. However, in very demanding applications where there is very heavy copper and a very tight pitch, the PTFE-ceramic film will conform and fill around the copper thus increasing the effective mass available to flow and fill the artwork.

DESIGN NOTE: In order to achieve adequate filling around etched features, the number of fastRise plies and ply thickness should be determined using the guidelines in Table 2. These guidelines are based on high layer count multilayer designs with stacked traces which create high and low pressure areas during lamination. The recommendations are conservative and other designs may realize satisfactory results using fewer or thinner bond plies. The flow properties are very dependent on the pressure used during lamination and the prepreg used. For low flow or cavity type applications, the prepregs with 4-5% flow should be used and low lamination pressures. For 2-3 oz copper filling, high flowing prepregs are necessary. 2 plies of prepreg may be necessary for 2-3 oz copper.

			High		Filling					
		Striplines	Layer	Between	cavities	Nominal		Adhesive		Microvia
	Speed	with 1.0	Count	subs	blind or	Carrier	Carrier	resin		Formation
	Board C	oz	stripline	with plated	buried	Film	Film	per	Flow	Foil
	part #	copper	PWBS	up copper	vias	(mils)		side	(%)	Lam
FR26-0025-60	2.2 mil	yes	yes	caution	no	1	smooth	Low	17	NP
FR27-0030-25		caution	no	no	no	2.3	rough	Low Flow	4	R
FR27-0035-66	3.4 mil	yes	yes	yes	yes	1	smooth	High	36	NP
FR27-0042-75		yes	yes	yes	yes	1	smooth	Highest	35	NP
FR28-0040-50		yes	yes	caution	no	2.3	rough	High	23	R
FR27-0045-35		yes	caution	caution	no	3	rough	High	13	R
FR27-0050-40		yes	yes	caution	no	3	rough	High	23	R
NR = not preferred but plateable, R = recommended										

Table 6 Choosing the Right fastRise Part Number

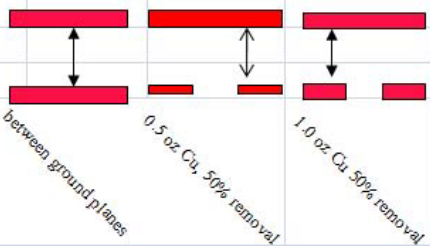
	Pressed Thickness (500 psi)			Nominal DK	Typical Flow	Number of Prepreg Plies Copper Weight (oz)		
	(mil)	(mil)	(mil)	(10 GHz)	(%)	(1/2)	(1)	(2 or 3)
FR26-0025-60	2.7	1.3	1.0	2.58	17	1	1	2
FR27-0030-25	3.5	2.1	1.8	2.74	4	1	NR	NR
FR27-0035-66	3.7	2.5	2.1	2.7	36	1	1	2
FR27-0042-75	5.16	4.0	3.5	2.73	35	1	1	1
FR28-0040-50	4.9	3.7	3.5	2.81	23	1	1	2
FR27-0045-35	5.8	4.6	4.2	2.73	13	1	1	2
FR27-0050-40	6.1	5.5	4.9	2.76	23	1	1	2
fastRiseDS	5				4	1	1	NR
fastRise77	5.5				4	1	1	NR
								

Table 7 Pressed Thicknesses and Design Guidance

fastRise is manufactured to a mass specification and not to a thickness specification. The final pressed thickness will depend on many factors including: (1) the mass of copper etched away (2) pitch (3) the presence of low pressure zones in the pwb (4) press pressure during multilayer lamination (5) the presence of features such as blind or buried vias that will draw thermoset resin away. fastRise will vary in thickness by no more than 9% when all of the lamination parameters have been established and they are consistent. Processing fastRise can be readily accomplished using standard epoxy circuit board processing methods coupled with either plasma or sodium treatment of the PTFE hole walls. The following process recommendations are based on circuit boards produced at several facilities. It should be noted that each board shop may have different equipment and methods that will require modifications to these recommendations. Drill and routing parameters and artwork compensation data are very dependent on circuit board thickness and design and should be adjusted based on the experience of each facility. The treatment of hole walls prior to plating must be done using a plasma or sodium treatment system.

fastRise

Refrigeration

fastRise is a non reinforced prepreg that is manufactured between release liners so that individual plies of fastRise do not stick together. The adhesive layer on the surface of the PTFE/ceramic film can be quite tacky especially for freshly manufactured material. It is recommended to refrigerate fastRise prior to lamination. Continuous refrigeration is always a good practice for storing prepregs as this will extend the shelf life. However, because fastRise can be quite tacky, fastRise should be refrigerated as close to 40ºF/4ºC as possible. fastRise will stiffen up and will separate from the release liners a
lot easier.

Lamination

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 [215.5°C] is within reach of most board shops.

AGC recommends the following press cycle for fastRise27 for most laminations:

- Vacuum Lamination Recommended
- Heat rise 3°F - 10°F / minute [1.5°C – 5.5°C] to 420°F [215.5°C] *
  - Flow window is 130°F [80°C] – 302°F [150°C]
- Maintain pressure at 73 psi [5 bar] until package reaches 100°F [37°C] then apply full pressure of 500 psi [31 bar] Hold (cure) for 1 hour
- Cool package under full pressure at < 6°F [3°C] / minute

Alternate press cycle for hybrids:

- Press at 390F for 3 hours, otherwise same as above

For fastRiseDS, fastRise77P, and very difficult flow and fill multilayers with fastRise27, AGC recommends the following press cycle:

- Vacuum Lamination Recommended
- Heat rise 3°F - 10°F / minute [1.5°C – 2.0°C] to 225 °F [121°C]
- Maintain pressure at 73 psi [5 bar] until package reaches 100°F [37°C] then apply full pressure of 500 psi [31 bar]
- HOLD AT 225°F for 60 minutes
- Increase temperature from 225°F [121°C] to 420°F [215.5°C] at 1.5 – 3.0°C / minute - Flow window is 130°F [80°C] – 302°F [150°C]
- Hold (cure) for 1 hour
- Cool package under full pressure at < 6°F [3°C] / minute

*For high layer count boards, best flow and fill has been seen at low heat-up rates. For low layer count multilayers, a low heat-up rate is less critical.

The as shipped prepreg cure is not very advanced. For best gap filling it is desirable to achieve the maximum amount of flow prior to the advancement of the resin. The earliest application of pressure will help, certainly below the melt of the thermoset (140°F [60°C]), coupled with a 3°F - 10°F [1.5°C -5.5°C] rate of rise. A cold start of the press is desirable. A hot start of the press will advance the cure of the resin before the best flow is achieved and the optimal amount of flow will not be achieved.

For lamination of 2 oz innerlayers a slow heat ramp up rate is suggested. Lagging or padding will help achieve a slower heat rise. Press pressure will affect the degree of flow and any voiding if present. Low pressure will yield less flow, more pressure more flow. A slow cool is necessary to avoid any issues associated with delamination. The hot press should be cooled to somewhere between 170-200°F before opening, although data suggests the press could be opened at 250°F. Transfer to a cold press should not be done until the hot press has reached the recommended temperatures.

Appendix

DOUBLE SIDED PTFE MATERIALS

Double Sided PTFE Laminate Microdrilling Parameters								
Drill Size (in)	Chipload (mil)	80,000 max		110,000 max		125,000 max		Retract Rate (in/min)
		Spindle Speed (rpm)	Feed Rate (in/min)	Spindle Speed (rpm)	Feed Rate (in/min)	Spindle Speed (rpm)	Feed Rate (in/min)	
0.0039	0.3	80,000	24	110,000	35	125,000	113	135
0.0059	0.5	80,000	39	110,000	55	125,000	125	145
0.0079	0.7	80,000	59	110,000	79	125,000	138	157
0.0098	1.0	80,000	79	110,000	106	125,000	157	197
0.0118	1.2	80,000	94	110,000	130	125,000	177	236
0.0138	1.3	80,000	106	110,000	146	125,000	189	276
0.0157	1.5	80,000	118	104,000	154	106,000	177	315
0.0177	1.7	80,000	134	92,000	154	92,000	169	354
0.0197	1.9	80,000	150	83,000	157	85,000	165	394

DOUBLE SIDED PTFE MATERIALS

	Double sided PTFE Drilling Parameters								
Drill Size (in)	Chipload (mil)	Spindle Speed (rpm)	Feed Rate (in/min)	Retract Rate (in/min)	Drill Size (in)	Chipload (mil)	Spindle Speed (rpm)	Feed Rate (in/min)	Retract Rate (in/min)
0.0217	1.8	80,000	140	400.0	0.0748	2.4	27,000	65	500.0
0.0236	1.8	77,500	141	400.0	0.0768	2.5	26,000	65	500.0
0.0256	1.9	75,000	140	400.0	0.0787	2.5	26,000	65	500.0
0.0276	1.9	72,500	135	400.0	0.0807	2.5	26,000	65	500.0
0.0295	1.9	70,000	130	400.0	0.0827	2.6	25,000	65	500.0
0.0315	1.9	66,000	125	400.0	0.0846	2.7	24,000	65	500.0
0.0335	1.9	62,000	120	400.0	0.0866	2.7	24,000	65	500.0
0.0354	2.0	60,000	120	500.0	0.0886	2.8	23,000	65	500.0
0.0374	2.0	57,000	115	500.0	0.0906	2.8	23,000	65	500.0
0.0394	2.0	54,000	110	500.0	0.0925	2.7	22,000	60	500.0
0.0413	2.1	51,000	105	500.0	0.0945	2.8	21,500	60	500.0
0.0433	2.1	48,000	100	500.0	0.0965	2.6	21,000	55	500.0
0.0453	2.1	45,000	95	500.0	0.0984	2.7	20,500	55	500.0
0.0472	2.1	43,000	90	500.0	0.1004	2.5	20,000	50	500.0
0.0492	2.0	42,000	85	500.0	0.1024	2.5	20,000	50	500.0
0.0512	2.0	40,000	80	500.0	0.1043	2.5	20,000	50	500.0
0.0531	2.1	39,000	80	500.0	0.1063	2.3	19,500	45	500.0
0.0551	2.1	38,000	80	500.0	0.1083	2.3	19,500	45	500.0
0.0571	2.2	36,000	80	500.0	0.1102	2.4	19,000	45	500.0
0.0591	2.4	34,000	80	500.0	0.1122	2.2	18,500	40	500.0
0.0610	2.3	32,000	75	500.0	0.1142	2.2	18,500	40	500.0
0.0630	2.3	32,000	75	500.0	0.1161	2.2	18,000	40	500.0
0.0650	2.3	32,000	75	500.0	0.1181	2.0	17,500	35	500.0
0.0669	2.4	31,000	75	500.0	0.1201	2.0	17,500	35	500.0
0.0689	2.3	30,000	70	500.0	0.1220	2.1	17,000	35	500.0
0.0709	2.4	29,000	70	500.0	0.1240	1.8	17,000	30	500.0
0.0728	2.5	28,000	70	500.0	0.1250	1.9	16,000	30	500.0

AGC
Your Dreams, Our Challenge

DOUBLE SIDED PTFE MATERIALS

Double Sided PTFE Microdrilling Parameters (Metric)								
Drill Size (mm)	Chipload (um)	80,000 max		110,000 max		125,000 max		Retract Rate (m/min)
		Spindle Speed (rpm)	Feed Rate (m/min)	Spindle Speed (rpm)	Feed Rate (m/min)	Spindle Speed (rpm)	Feed Rate (m/min)	
0.10	7.6	80,000	0.61	110,000	0.89	125,000	1.00	2.99
0.15	12.5	80,000	1.00	110,000	1.40	125,000	1.60	4.49
0.20	18.8	80,000	1.50	110,000	2.00	125,000	2.30	5.98
0.25	25.0	80,000	2.00	110,000	2.70	125,000	3.10	7.48
0.30	30.0	80,000	2.40	110,000	3.30	125,000	3.70	8.98
0.35	33.7	80,000	2.70	110,000	3.70	125,000	4.00	10.47
0.40	37.5	80,000	3.00	104,000	3.90	106,000	3.90	11.97
0.45	42.5	80,000	3.40	92,000	3.90	92,000	3.90	13.47
0.50	47.5	80,000	3.80	83,000	4.00	85,000	4.00	14.96

DOUBLE SIDED PTFE MATERIALS

Double sided PTFE Drilling Parameters (Metric)								
Drill Size (mm)	Chipload (um)	Spindle Speed (rpm)	Feed Rate (m/min)	Retract Rate (m/min)	Drill Size (mm)	Chipload (um)	Spindle Speed (rpm)	Feed Rate (m/min)
0.55		76,000	3.8	10.2	1.90	81.8	22,000	1.8
0.60		69,000	3.6	10.2	1.95	81.8	22,000	1.8
0.65		64,000	3.5	10.2	2.00	81.0	21,000	1.7
0.70		60,000	3.4	10.2	2.05	81.0	21,000	1.7
0.75		56,000	3.2	10.2	2.10	80.0	20,000	1.6
0.80		52,000	3.1	10.2	2.15	80.0	20,000	1.6
0.85		49,000	3.0	10.2	2.20	80.0	20,000	1.6
0.90		46,000	2.9	12.7	2.25	80.0	20,000	1.6
0.95		44,000	2.9	12.7	2.30	80.0	20,000	1.6
1.00		42,000	2.9	12.7	2.35	80.0	20,000	1.6
1.05		40,000	2.8	12.7	2.40	80.0	20,000	1.6
1.10		38,000	2.7	12.7	2.45	80.0	20,000	1.6
1.15		36,000	2.7	12.7	2.50	80.0	20,000	1.6
1.20		35,000	2.7	12.7	2.55	80.0	20,000	1.6
1.25		34,000	2.7	12.7	2.60	80.0	20,000	1.6
1.30		32,000	2.6	12.7	2.65	80.0	20,000	1.6
1.35		31,000	2.6	12.7	2.70	80.0	20,000	1.6
1.40		30,000	2.5	12.7	2.75	80.0	20,000	1.6
1.45		29,000	2.4	12.7	2.80	80.0	20,000	1.6
1.50		27,000	2.3	12.7	2.85	80.0	20,000	1.6
1.55		26,000	2.2	12.7	2.90	80.0	20,000	1.6
1.60		26,000	2.2	12.7	2.95	80.0	20,000	1.6
1.65		25,000	2.1	12.7	3.00	80.0	20,000	
1.70		24,000	2.1	12.7	3.05	80.0	20,000	
1.75		23,000	2.0	12.7	3.10	80.0	20,000	
1.80		23,000	1.9	12.7	3.15	80.0	20,000	
1.85		23,000	1.8	12.7	3.18	80.0	20,000	

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.

AGC Inc.
agc-ml.info-maltimaterial@agc.com