



PROCESSING GUIDELINES

Laminate: SCGA-500 Series

(SCGA-500 GF220, SCGA-500 GF255, SCGA-500 GF265, SCGA-500 GF300)

High Frequency PTFE Material



This process guide is based on the IPC standard IPC-4103, and Shengyi makes some adjustment and generates it based on the product features, so as to help our customers better use the SCGA-500 series PTFE products.

1. Storage Condition

1.1 Laminate core

1.1.1 Store Condition

- Place on platform or on the appropriate frame. Avoid heavy pressure or leaning against wall to prevent sheet deformation, which may impact the subsequent PCB processes.

1.1.2 Storage Environment

- Sheets should be stored in ventilated, dry, at room temperature under environment control, avoiding direct sunlight, rain and avoid erosion of corrosive gas (stored environment directly affect the quality of material).
- For double-sided copper-clad laminates (cores), shelf life is suggested up to two years in suitable storage condition. Minimize moving the products to avoid surface scratches.

1.1.3 Operation

- Be aware that the product is PTFE based material and has softer mechanical futures compared to FR-4 material, and handle with extra care.
- Wear clean gloves and operate the products with caution. Avoid collisions and sliding, which will damage the copper foil. Do not use bare hands to operate the products as it will cause contamination to copper foil surface.

2. PCB Processing Guide

2.1 Cutting

- Sawing machine is suggested as the first choice for cutting the product and shearing machine can be used as the secondary choice. Roller cutting is not recommended as it may cause edge delamination.

2.2 Core baking

- Generally baking is not recommended for this product. Core baking may be carried out based on actual usage situation. For baking after cutting, wash the panels using high-pressure water after cutting then bake the panels. Baking condition is suggested 130°C/1 ~ 2h. Avoid direct contact of the cores and heater and do not introduce resin powder onto the surface in the cutting process, which may cause etching defects.

2.3 Pretreatment

- Do not use any mechanical brushing/scrubbing during the whole PCB process. Try to use chemical cleaning for pretreatment process.

2.4 Etching

- After inner layer etching process, lamination process should be done as soon as possible, not later than 4 hours.



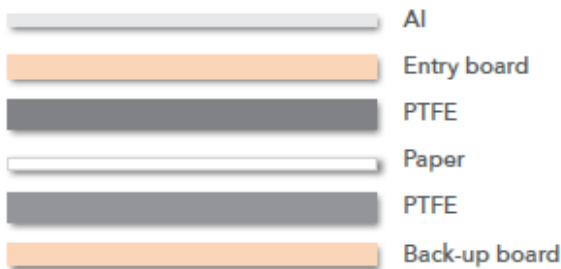
2.5 Lamination

- Press cycle should be determined according to the prepreg used.

2.6 Drilling

- Entry and back-up board: Use Aluminum board + phenolic board or similar rigid for entry. Use phenolic composite panel or similar rigid board for back-up.
- Stack-up quantity: As PTFE is very soft, the stack-up quantity is advised as less as possible to avoid hole shifting. When base board thickness $\leq 30\text{mil}$, recommend 2 panels/stack; When base board thickness $\geq 60\text{mil}$, recommend 1 panel/stack. For 2 panels/stack, please use white paper in-between for isolation, to avoid scratches.
- Stack-up construction: Advised: Aluminum board + Phenolic board + PTFE (white paper + PTFE) + Phenolic board. For hybrid multilayers combined with PTFE and FR-4, it is recommended to put PTFE at the bottom and FR-4 on the top.

Stack-up sketch



- Drill bit and hit counts: For PTH holes, new drill bits are recommended and do not use re-sharped ones. Max hit count 300-500. For Non-PTH holes, re-sharped drill bits are usable and re-sharped time depends on drilling quality and not more than 3 times.
- Drilling parameters
 - PTFE drilling parameters, for reference only.

Diameter (mm)	Speed(Krpm)	Infeed(IPM)	RTR(IPM)	Maxhit
0.2	70	70	500	300
0.5	50	75	500	400
0.8	30	60	500	500
1.0	25	50	500	500
2.0	20	40	500	500
3.0	20	40	500	400

Remark: Drilling parameters depend on different drilling machine, types of drill bit and hole diameter, and there's relationship among spindle speed, in-feed and return speed, so evaluation for specified



drilling parameters is recommended.

- PTFE and FR-4 hybrid drilling parameters, for reference only.

Diameter mm	Speed(Krpm)	Feed(IPM)	RTR(IPM)	Maxhit
0.2	120	50	500	300
0.5	77	77	500	400
0.8	55	83	500	500
1	42	83	500 5	00
2	25	50 5	00	z500
3	25	50	500	400

Remark: For hybrid combined with different FR-4 and construction, the drilling parameters should be different and revised correspondingly.

2.7 Core baking after drilling

- Recommended baking condition: 130°C/1 ~ 2h. Avoid direct contact of the cores and heater.

2.8 Desmear

- PTFE can't be dissolved by solvents and non-chemical method is preferred for smear cleaning.
- We recommend taking plasma treatment. For better cleaning, ultrasonic rinse before plasma is preferred and then bake at 110-120°C/60-90min.

- Plasma parameter 1:

Step	O2	N2	CF4	H2	Pressure (PSI)	RF (w)	Flow Rate	Seg Time (min)
1	80	10	10	0	250	4000	2.5	15
2	0	20	0	80	250	4000	2.5	45

- Plasma parameter 2:

O2	N2	CF4	Mode	Pressure	RF Watts	Flow Rate	Seg Time
80	20	0	V	250	6500	3	30
80	10	10	P	240	6500	3	10
100	0	0	P	250	6500	2.5	5
0	100	0	P	250	6500	3	30

2.9 PTH

- The desmear step in PTH process is not necessary, can directly start from the rinse step. If twice PTH needed, the 2nd PTH can start from pre-dipping step.

2.10 Outer layer AOI

- The PTFE surface will gradually become smooth as time goes by, therefore recommend solder mask process as soon as possible after AOI and do not stay for a long time at this process. If needed, less



than 8 hours is advised.

2.11 Solder mask

- The bonding between solder mask and base material is only based on mechanical strength but not chemical bonding, so mechanical brushing before solder mask is not recommended in case of destruction of original surface roughness, which might weaken the bonding strength and cause solder mask peel-off problem.
- If solder mask peel-off detected, it's advised to remove solder mask and go through plasma process again to increase the surface roughness, which will strengthen the bonding between solder mask and base material.
- Low surface energy of PTFE material causes weak bonding strength to solder mask. It is recommended to decrease solder mask thickness, and increase UV pre-cure energy level(recommend above 10 level), increase low-temperature post-cure time. Select suitable solder mask ink can also help strengthen the bonding.

2.12 Surface finishing

- For antenna application, immersion tin surface finishing is recommended. Immersion silver or gold are also available for other applications.
- Please avoid HAL surface finishing, if necessary, we recommend parameters as below:
 - 1) Bake production boards at 150°C/2-3H before HAL.
 - 2) HAL temperature should lower than FR-4 requirement and avoid sharply heat-up and cooling during processing.
 - 3) Peak HAL temperature is recommended lower than 255°C.

2.13 Routing& Punching

- Routing is recommended. Punching is not recommended.

2.14 Packaging

- To prevent moisture effect on the heat resistance of base material, suggest baking finished boards at 130°C/3-5h before packaging.
- It's recommended to use vacuumed aluminum pack.

3. PCB Soldering

3.1 Shelf life of PWB

- 3 months with aluminum packaging protection.
- Baking at 130°C/1-3h is recommended before assembly.

3.2 Reflow

- Suitable for lead free reflow process.

This process guide is for reference only. Should you have any questions, please feel free to contact us. Shengyi will support you with prompt and effective service.