AGC
Your Dreams, Our Challenge

N9000-13RF

General Processing Guidelines

PTFE Performance Blended Substrates

N9000-13RF is a next generation PTFE Performance Blended PCB material which combines the RF electrical properites of PTFE with the competitive performance features of a high speed epoxy, i.e. N4000-13.

Material Handling & Storage

Store laminate and bond ply material ina flat orientation and in a dry environment (≤50% RH).
Do not bend, scratch or dent laminate.

Drilling

N9000-13 RF materials should be drilled using highly polished carbide tools. The use of repointed tools is not recommended. Panels can be drilled in stacks based on total thickness. The total stack height, including entry and backup materials, should not exceed 70% of the effective flute length of the drill bit. For a multilayer PCB, 1 panel per stack is preferred. The use of hard phenolic, rigid entry .020-.032" (0.5 mm - 0.8 mm) and backup .060-.080" (1.5 mm – 2.0 mm) material is recommended. PTFE laminates are much softer than FR4 laminates. The edges of the holes will be distorted if entry and backup materials are not used during drilling. Aluminium or aluminium-clad entry and backup materials are not recommended for N9000-13 RF materials, as aluminium burrs have a tendency to damage the soft PTFE during the machining process. The following drill parameters are recommended as a beginning point to develop specific process parameters.

Drill bit (mm)	INFEED (IPM)	SPEED (KRPM)	RTR (IPM)	Chip load (mil)	Hit Max
0.20	73	73	300	1.0	500
0.25	89	68	300	1.3	500
0.30	85	60	300	1.4	500
0.35	83	55	300	1.5	500
0.40	87	52	300	1.7	500
0.50	96	48	400	2.0	500
0.65	74	37	400	2.0	600
0.75	64	32	400	2.0	600
1.00	48	24	400	2.0	600
1.25	40	20	400	2.0	600

Hole Cleaning/Deburr

Use proper entry and exit materials to minimize burr formation. If the burrs are small, use a hand pumice scrub followed by high pressure rinsing to remove debris from the holes. Mechanical deburr or scrubbing to remove burrs is not recommended. All loose debris in the holes should be removed prior to plasma treatment.

AGC
Your Dreams, Our Challenge

Bake
If the PCB is a hybrid with N4000-13, then bake after hole cleaning. Bake condition could be 3-4hours @150°C.

Desmear
PTFE is chemically inert. There is no material available that can chemically remove drill smear from a hole in PTFE. The only way to remove the smear is to use a larger drill bit and cut the smear out of the hole. Plasma processes typically used to desmear epoxy-based boards (O_2 / CF_4) appear to have limited effectiveness when treating PTFE surfaces. By contrast, NH_3 or (70% H_2 / 30% N_2) blends have been found to yield good results with PTFE as well as sodium etching.

The typical plasma cycle contains three steps: An O_2/N_2 burn to raise the product temperature, a main process cycle, and an O_2 burn to remove ash and residual unreacted resin components. Baking at 30-60min@120°C before Plasma is recommended.

Step	O2	N2	CF4	H2	Pressure	RF	Seg Time (min)	Remark
1	80%	20%			250Torr	8000W	30	
2	80%	10%	10%		250Torr	4000W	10-45	For FR4
3	100%				250Torr	4000W	10	
4		30%		70%	250Torr	4200W	10-30	For PTFE

Metallization
Skip the chemical desmear and the panels should be processed through metallization within 8 hours after plasma. If the hold time is exceeded, re-activate the PTFE using step 1 and step 4 from the list above.

Imaging
The dimensional stability of N9000-13 RF material is excellent. The dimensional change should be <0.5 mils/inch (0.005 mm/cm), depending on the construction and process.

Solder Mask
If the PTFE is in the outer layer, the solder mask should be applied within 24 hours after etching; the tooth of the copper provides mechanical bonding of the mask to the PTFE surface. If the hold time is exceeded, re-activate the PTFE using step 1 and step 4 from the list above.

Reflow/HASL
Bake prior to re-flow or hot air solder leveling.

Developing/Etching/Stripping
Due the low modulus of the PTFE materials, frames and leaders may be required when processing thin laminates through some conveyorized equipment. Standard etchants can be used.

Routing
Routing, steel rule die cutting, waterjet cutting, and laser cutting can be used depending on required edge quality and cutting tolerances. In the case of routing, standard carbide (helical two flutes or four flutes) routers are recommended. Copper foil should be etched away from the routed edge to prevent burring. For edge plating, follow the hole plating recommendations above.

Appendix

*If the out layer of the PCB is FR-4, not PTFE, then the entry and back-up materials for drilling can be aluminum or aluminum-clad. In this case, mechanical deburring can be applied.

*If the PCB is a multilayer hybrid with the N4000-13 series, the peak temp. for lamination should be less than 195°C, and the cure time should be less than 110 min.

The above processing guides are recommendations only and intended for general review purposes. Process adjustments may be required to achieve optimum results in your specific manufacturing environment. For more detailed processing information, please contact with the AGC engineer or sales representative.

agc-ml.info-maltimaterial@agc.com