The T-lam System

Single Layer IMpcb Fabrication Guideline

Part III: Directions for T-lam IMpcb Single Layered Material Panel Processing

Samples sent in twenty-four hours. Solutions that change a world.
4707 Detroit avenue* Cleveland, Ohio 44102* 1-888-246-9050*www.thermagon.com
*Interface Materials*Gap Fillers*Phase Change Materials*Encapsulants*Circuit Board Pre-Pregs and Laminates*

Single Layer IMpcb Fabrication Guideline

Part III: Directions for T-lam IMpcb Single Layered Material Panel Processing

Paul J. Klouda

This document will provide printed circuit board fabricators information about processing single layered **T-lam IMpcb** material. Effectively, **T-lam IMPCB, DSL, T-lam/Hybrid** circuits all process in condition similar to single sided print and etch, DSPTH and Multilayered FR-4 constructions. There are some process differenced, most concerning the protection of the base metal and handling the over all weight and thickness of the finished material. We will address these issues in the following sections of this document.

.

1.0 Raw Sheet and/or Panel Material Handling
- 1.1 T-preg material
- 1.2 T-lam IMpcb material

2.0 Panel Prep
- 2.1 Tooling/Registration
- 2.2 Scrubbing
- 2.3 Chemical Cleaning
- 2.4 Base Metal Protection

3.0 Imaging
- 3.1 Dry Film Application
- 3.2 Wet Film Application
- 3.3 Screened Image Application

4.0 Wet Processing
- 4.1 Etching
- 4.2 Stripping
- 4.3 Post Etch/Strip Clean

5.0 Solder Mask Application
- 5.1 LPISM – Screen, Spray or Curtin Coat
- 5.2 Thermal Solder Mask
- 5.3 UV Cured Solder Mask

6.0 Second Step Drill or Punch
- 6.1 Drill
- 6.2 Punch

7.0 Finishing Operation
- 7.1 HASL
- 7.2 OSP
- 7.3 Tin
- 7.4 Ni/Au

8.0 Fabrication
- 8.1 Rout
- 8.2 V-score
- 8.3 Punch

9.0 Hipot Testing
- 9.1 Voltage, Time and Test Parameters

10.0 Packing and Handling for Shipments

1.0 Raw Sheet and/or Panel Material Handling

Handling and storage of T-preg materials is critical to the successful processing and final quality of the fabricated IMpcb. This section covers the basic procedures for proper handling and storage of both T-preg and T-lam IMpcb and T-lam DSL.

1.1 T-preg Material

T-preg (B-staged pre-preg) material is a room temperature stable (at or under 23 C), ceramic filled epoxy sheet product. The material is supplied with a protective release liner, which must be removed prior to any lamination process. T-preg is supplied in thickness of .006" to .012" in 18" x 24" sheets. The use of latex or synthetic rubber gloves during handling is recommended to eliminate material contamination from exposure to skin.

T-preg material has a shelf life of 6 months or longer if stored between 5 – 20 C at ~50% RH.

1.2 T-lam IMpcb Material

T-lam IMpcb (Insulated Metal printed circuit board) material is a single layered laminate with copper foil 1oz/ft to 6 oz/ft, T-preg dielectric layer .006" to .012" thick and base metal layer usually consisting of either aluminum or copper .040" - .125" thick. The material is supplied in 18" x 24" panels with or without etch protect masking on the base metal material. If base metal masking is requested the masking material must be removed prior to any processing done at temperatures greater than 105 C for 10 minutes. Ideal 9145 or 9148 tape is recommended as an etch resist for the base metal material.

When processing panels of T-lam IMpcb care must be taken to avoid contamination of the copper foil prior to imaging. The use of gloves when handling T-lam IMpcb material is recommended.

2.0 Single Layer IMpcb Fabrication Guidelines

Single layered metal based boards or IMpcb's (Insulated Metal printed circuit boards) have the widest acceptance amongst power supply, power module and motor control manufacturers. The following outline details the individual steps in fabricating an IMpcb.

2.1 Tooling and Registration Preparation

Preparation of the panel to allow for imaging and subsequent processing can be accomplished in a few methods.

Drilling or punching of registration or tooling holes can be performed prior to or post imaging/etching. For drilled tooling holes the feeds and speeds are different from FR-4. This accommodates the base metal on the panels. Feed rates of 0.001" – 0.0015"/rev with spindle speeds of 24,000 – 30,000 rpm. Stack heights of 1 – 4 panels are possible depending upon the material thickness. Punching tooling holes requires a punch and die set that will withstand the base metal characteristics. Contact a tool and die manufacturer for specific details on tool construction.

The next step, panels are cleaned by mechanical scrubbing, chemical cleaning or high impingement slurry such as pumice or aluminum oxide. A water break test of 30 sec minimum hold time will confirm a clean foil surface for imaging.

2.2 Imaging

2.2.1 Dry Film Application

1.3 to 2.5 mil aqueous developable dry film photo resist can be applied to cleaned T-lam IMpcb panels. Standard hot roll or cut sheet laminators are acceptable. Slower than normal lamination speeds may be required in order to bring the metal-based material up to proper lamination temperature.

2.2.2 Wet Film Application

Some liquid or wet film photo resists are available. If wet film photo-resist is chosen follow manufacturers suggested application method. Apply material to a cleaned panel. Double-sided coating can be used to protect base metal during etching. Follow manufacturers recommended tack dry, exposure, develop and post cure guidelines.

2.2.3 Developing

Most dry film and wet film products develop in a mild caustic solution. Protection of the base metal is not required for developing solutions. However, if applied the base metal etch mask will withstand developing chemistry. Follow manufacturers recommended guidelines.

2.2.4 Screened Image Application

There are 2 primary types of screen printable etch resists, thermally curable and UV curable. Currently, most fabricators are using UV curable etch resist inks. Panels must be clean and free from fingerprints and dirt prior to screening. Preparation of the image screen should be handled per stencil manufacture instructions. Curing should follow manufacturers recommended guidelines.

If a base metal etch protection coating is required, it is possible to use screen-printed etch resist ink to accomplish this task. Panels must be clean and free from fingerprints, dirt and contamination prior to print cycle. Preparation of image screen should leave a slight border around perimeter of panel approx .050”. Curing should follow manufacturers recommended guidelines.

3.0 Wet Processing

Wet processing includes etching, stripping, and post etch cleaning. Each process requires special attention to maintain quality, throughput and consistency. Refer to manufacturers directions when using any chemistry. The methods listed below are a guideline and should not over ride specific recommendations from chemistry suppliers without prior communication.

3.1 Etching

There are 3 primary ways of etching IMpcb material, spray, immersion and paddle (splash). We will concentrate on spray techniques as the process and equipment is more common with fabricators. Vertical or horizontal equipment will work. There are main 3 chemistries involved with etching - alkaline, cupric chloride and ferric chloride.

Each etching system has advantages and disadvantages. Alkaline systems work well with foil up to 3 oz. The etch rate is fast and the results uniform. Using alkaline etch on greater than 3 oz foil creates 2 problems. 1) The aqueous photo resist begins to break down during etch and 2) The undercut/over etch begins to significantly effect the line width and spacing.

Cupric chloride is an acidic etching chemistry that performs well and does not attack aqueous photo resist. The major drawback with cupric chloride is it’s slow etching rate. However, even with a slow etch rate the overhang, undercut or over etch geometry is minimal. Rejuvenation of cupric chloride is possible check with the etchant supplier for equipment and procedures.

Ferric chloride is also an acid etching chemistry that performs well and does not attack aqueous photo resist. Ferric is also slower than alkaline based chemistry but do to the economical advantages can be a good choice for etching. Overhang, undercut and over etch characteristics are similar to cupric chloride.

3.2 Stripping

Stripping the etch resist off the panels can be accomplished by 2 primary processes. The first method is rack, dip and rinse. Due to the limitations of this manual process and the possibility of excessive attack on the base metal this is not the preferred method.

In line conveyorized stripping is the preferred method. Depending upon the resist material used the stripping solution makeup; temperature and dwell time in the spray chamber will vary. Consult your resist supplier for exact specifications.

3.3 Post Etch Clean

This procedure is a pre-mask preparation. Mechanical and or chemical cleaning of the copper surface is required to prepare the surface for solder mask application. A 320 grit bristle brush or compressed fiber brush will work adequately. Pumice or another oxide slurry will also work if available. A good 30 sec water break test will confirm proper surface cleanliness.

4.0 Solder Mask

Solder mask material has been used for many years in printed circuit board production. In the case of IMpcb products the uniformity, cleanliness overall cosmetic appearance are critical factors. Some customers use the mask as a conformal coating and an electrical insulation area. While most solder mask material exhibit favorable properties they are not specifically intended for this use. There are 4 primary

4.1 Liquid Photo-Imageable Solder Mask

LPISM is the most common version of coating boards prior to finishing. Application by screening, spraying or curtain coating can give satisfactory results. Thicker copper weights may require double passing to ensure complete coverage and adequate thickness at the knee of the traces and land areas. Contact your ink manufacturer for exact process details.

4.2 Thermal Cured Solder Mask

Thermal cured solder masks have been used for the longest time. Due to the advancement in LPISM and UV cured masks, thermal masks are not in as great of demand. However, they still perform very well if the proper surface treatments of the copper surface and correct curing schedules are followed. The most common thermal mask products are either a one part or two-part mixture and pot life is a consideration. Follow manufacturers recommended process directions for best results.

4.3 UV Cured Solder Mask

In high volume single sided board manufacture, in-line UV mask application is still widely accepted. Unless the copper surface is treated properly lower mask adhesion is the principle holdback for use in IMpcb manufacture. UV solder masks also produce a “halo” effect or die separation at the annular ring or pad edge in surface mount products. This clear area of UV material can appear as de-wetting or mask bleed. The appearance is strictly cosmetic, but can have negative consequences with QA inspections. This is not a recommended process for IMpcb products.

4.4 Dry Film Solder Mask

Dry film solder mask has been used for the past 12 – 15 years on higher density circuitry due to the increased definition and relative ease of application. In some applications this may be an appropriate mask choice, but for general applications this method is not recommended.

5.0 Second Step Drill or Pierce Punching

Adding through holes can be accomplished at this time. Drilling or punching holes are the 2 main methods of completing this operation. This operation may need to be performed after finishing to eliminate the possibility of chemistry contamination, specific procedures will dictate this adjustment.

5.1 Second Step Drilling

Using a feed rate of 0.001” – 0.0015”/rev with spindle speeds of 24,000 – 30,000 rpm. A strong vacuum is required to extract chip and debris from the hole and to cool the drill bit properly. Stack heights of up to 4

panels can be used if the drill bit is sharp and no burring is present. Entry material and hard backup board is required to minimize burring.

5.2 Pierce Punching

Similar to tooling or registration holes a well-constructed punch and die set is required for proper hole quality. Refer to local tool and die manufacturer for specific details of tool construction and use.

6.0 Finishing Operations

In order to prepare the IMpcb for component placement and assembly the exposed copper surfaces must be treated to ensure adequate solderability, proper wire bond strength and overall manufacturability. A variety of coatings, plated metals and conversion treatments are available. We will discuss 4 of the choices: HASL, OSP, Sn and Ni/Au.

6.1 HASL

HASL or Hot Air Solder Leveling is currently the preferred choice of finish on IMpcb's. Either horizontal or vertical application is acceptable. At this point the base metal mask if used must be removed. Fluxing, preheating, application and post cleaning all follow standard process methods. An increased dwell time in the solder pot may be required to allow the panel to reach temperature. This may reduce machine cycle frequency to allow for solder pot temperature recovery. The IMpcb's act as large heat sinks requiring the increased dwell time. Prior to post cleaning the coated panel may have a black or grayish smut coating on the base metal side of the panel. It can be removed by an aggressive aqueous cleaning cycle, with mechanical brushing as an option.

6.2 OSP

When using an OSP or Organic Solderability Protection the base metal may require masking to protect the OSP chemistry from contamination. OSP's have shelf life limitations ranging from 3 weeks to 3 months. Care in handling the coated boards is required. Some OSP products allow exposure of metal without degradation to the chemistry. Contact chemistry supplier for process details.

6.3 Immersion or Electroless Tin Coatings

Similar handling procedures to OSP's are used when applying immersion or electroless tin coatings to IMpcb parts. The base metal must be covered to protect the tin chemistries from being contaminated. As with OSP coated parts Tin coated parts have shelf life limitations ranging from 1 month to 6 months. Care in handling the coated boards is required. Refer to chemistry supplier for process details.

6.4 Ni/Au Coating

Ni/Au or Nickel/Gold coatings are popular with wire bond applications. Ball bond, ultrasonic bond, thermosonic bond and wedge bond techniques all require a Ni/Au bonding surface. Most hard low to med phosphorus nickels work very pure soft gold are mostly used for the top coating. Each customer will have specific requirements and you will need to contact the chemistry supplier to define the correct plating baths for the operation.

7.0 Fabrication

This section will cover the aspects and procedures to singulate, palletize and finish the IMpcb's into shippable units. We will discuss the 3 primary means of fabrication: routing, v scoring and punching.

7.1 Routing

Routing is used for low to medium volume quantities. The proper tool selection along with a mist spray of water-soluble cutting fluid is required for best results. The use double fluted carbide router bit designed for metal removal is essential. 0.093" – 0.125" diameter with a spindle speed of 18,000 – 24,000 rpm, a chip load of 0.0015" – 0.003"/rev and a table travel of 22" – 50"/minute is recommended. Megatool, Tulon and other

carbide tool suppliers provide suitable products. Adequate vacuum and cutting fluid is required to maintain tool life and minimize burring. Stack heights of 1 – 4 panels are possible.

7.2 V-scoring

V scoring is a very popular and economical method of fabricating IMpcb's in product that has square non-radius corners. Low to high volume quantities is possible with v scoring. Minimum set up time and fast throughput make this a logical choice of fabrication. Both singulation and palletizing is accomplished with V-scoring. Carbide or diamond-tipped blades are acceptable. The use of diamond tipped blades requires special care and handling to ensure long tool life. V-scoring using 24 - 55 tooth zirconium nitride coated carbide steel blades with a feed rate of 10 – 25/fpm at a blade speed of 2500 – 6000 rpm. Water-soluble cutting fluid is recommended. Multiple pass increasing the depth of score until a .008" - .012" web remains will reduce the burrs.

7.3 Punch and Die

Single cavity or multiple cavity dies work with IMpcb material. Specific tooling instruction is available through local tool and die manufacturers. The fillers in T-lam IMpcb material will not wear the die and actually act as a lubricant.

8.0 Hipot Testing

Due to the nature of the metal base material on IMpcb parts, hipot testing to ensure isolation of the circuitry to base metal is required by most end users. There are a variety of test methods, fixture constructions and equipment types to accomplish this task. This portion of the guide will only cover the basics. Customer specifications will dictate the actual test requirements.

8.1 Hipot testing is performed using a modified spring-loaded bed of nails fixture. Using net list data and only contacting 1 or 2 points per net a fixture is built to connect all the foil circuitry on the IMpcb. A test voltage of 500 – 2500 VDC for up to 1 minute is applied to the circuitry through a low current isolated supply. The base of the IMpcb is connected to ground. Minimal leakage current may be detected during ramp up to final test voltage. A 5 – 10 sec ramp may be required to eliminate false failures triggered by capacitance build up in the IMpcb dielectric. A 10 sec – 1minute dwell may be required on each part. See customer specifications for exact test parameters and requirements.

8.2 Fixturing for hipot testing can be accomplished by building a single point per net spring loaded (pogo-pin) fixture. Please refer to hipot fixture detail provided separately. Other methods that achieve 100% connection of all land areas are acceptable.

9.0 Packaging

Packaging of the finished IMpcb's is important to minimize chaffing, scratching and abrading the surface of the boards.

9.1 Interleaving with a low sulfur release sheet and vacuum sealing stacked boards is a good method of packaging IMpcb's. Also individual wrapping and bubble packaging can be an alternative. It is critical to not stack boards solder coating to aluminum without separation or slip sheeting to eliminate galvanic reaction to the dissimilar metals. Loose stacking of boards does not present as great of a problem as tightly packed sealed stacks of boards.