The T-lam System

Multi Layer IMpcb Fabrication Guideline

Part IV: DSL, DSL/Hybrid and Multilayer IMpcb Fabrication Guidelines

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*

Multi Layer IMpcb Fabrication Guideline

Part IV: DSL, DSL/Hybrid and Multilayer IMpcb Fabrication Guidelines

Paul J. Klouda

This document will provide printed circuit board fabricators information about processing DSL (Double Sided Laminate), DSL/Hybrid (Double Sided Laminate/Hybrid FR-4 & Mixed technology) and Multilayered layered T-lam IMpcb material and pcb's. Effectively, T-lam IMPCB, DSL, T-lam/Hybrid circuits all process in conditions similar to single sided print and etch, DSPTH and Multilayered FR-4 constructions. There are some process differences, 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 Sheet Material and Panel Handling

1.1 T-preg Handling Procedures
1.2 T-lam IMpcb Handling Procedures
1.3 T-lam DSL Handling Procedures
1.4 Base Metal Handling Procedures

2.0 Panel Prep

2.1 Tooling/Registration
2.2 Compensation/Scale Factor

3.0 1st Step Drilling

3.1 Stack and Pin
3.2 Feeds and Speeds
3.3 Hit Count and Bit Wear
3.4 Post Drill Cleaning

4.0 PTH Process

4.1 Pre Plate Cleaning/Desmear
4.2 Electroless Copper
4.3 Direct Metalization
4.4 Flash Cu Plate

5.0 Primary Imaging

5.1 Dry Film Application
5.2 Wet Film Application
5.3 Screened Image Application
5.4 Developing

6.0 Wet Processing

6.1 Electrolytic Plating
6.2 Dry Film Stripping
6.3 Etching
6.4 Etch Resist Stripping
6.5 Post Etch Baking
6.6 Copper Adhesion Promotion

7.0 Lamination Cycle

7.1 Material and Base Metal Preparation
7.2 Lay Up
7.3 Press Procedure
7.4 De-Booking and Post Press Bake

8.0 Post Lamination Drilling/Punch and Second Step Drilling

8.1 Post Lamination Registration
8.2 Surface Cleaning
8.3 Spot Face Drill
8.4 Post Lamination Punch

9.0 Solder Mask Preparation and Application

9.1 Panel Cleaning
9.2 Mask Application
9.2.1 LPISM –Screen, Spray or Curtin Coat
9.2.2 Thermal Cure
9.2.3 UV Cured Screen Print

1.0 Sheet Material and Panel 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 DSL Material

T-lam DSL material is supplied in sheet/panel form usually 18” x 24”. The material should be handled in the same manner as any thin core laminate product. Support in the form of trays or racking is necessary to maintain flatness and to minimize fracture of the epoxy dielectric.
Storage of DSL in a dry 40 – 60 %RH and cool 15 – 23 C room is sufficient to maintain the material indefinitely.

1.3 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.

1.4 Base Metal Handling Procedures

Follow manufactures recommended storage procedures for aluminum and copper base metal sheets. Handle base metal sheets using cloth or leather gloves to minimize cuts and abrasions from sharp edges. Avoid direct skin contact to minimize oil and contamination of metal surface prior to pre lamination cleaning and surface preparation.

2.0 Panel Prep

Proper set up for tooling and registration is required prior to any manufacturing steps. This chapter outlines and describes the methods for successful set up.

2.1 Tooling / Registration. 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 20,000 – 60,000 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.

2.2 Compensation/Scale Factor. Artwork scaling may be required to allow for material movement during lamination and etching operations.

Here is a compensation factor for our T-lam DSL material. The information has been calculated for an 18” x 24” panel. This is a guideline based on 3 oz copper foil on both sides of the panel. When working with different foils weights you may need to increase or decrease the compensation based on the copper weight of each layer and percentage of copper fill. The compensation factor is applied to the artwork prior to photo plotting and does not include etch compensation for the copper weight. The core thickness for this calculation is 0.008" T-preg 1KA material.

X Axis 18" dimension the total compensation is 0.007" or 1.00038" factor per inch.
Y Axis 24" dimension the total compensation is 0.012" or 1.0005" factor per inch.
The difference in the X -Y axis can be attributed to the 104 cloth used in the T-preg.

3.0 First Step Drilling

Drilling T-lam material is accomplished through standard drilling techniques. The details included are a guide to some of the critical operations.

3.1 Stack and Pin. Orient the T-lam DSL panels to match the drill program. Stack and pin per standard thin clad laminate procedures. Use either foil board or composite backup material and appropriate entry material. Multiple panels can be stacked based upon drill condition and flute length.

3.2 Drilling of T-lam DSL is accomplished using standard feeds and speeds for FR-4 material. Adequate vacuum and pressure foot is required to maintain a clean hole and debris removal.

3.3 Hit Count and Bit Wear. The drilling of T-lam DSL can produce slightly higher bit wear and hit counts should be managed accordingly. Typical hit counts for hole diameters under 0.093” should be between 1000 – 1500. Hit counts for hole sizes larger than 0.093” may run from 500 – 1000. Last hit coupon holes are recommended and review of tools during initial trials is recommended.

3.4 Post Drill Cleaning. Any of the acceptable methods for cleaning/deburring thin core materials will yield satisfactory results on T-lam DSL panels. Follow equipment manufacture guidelines for set up and maintenance of cleaning process.

4.0 PTH Process

Applying plating in through holes is critical to the success of any double layer or multilayer board. T-lam material is compatible with most process chemistry. Hole condition and panel surface treatment are important to the success of any plating application.

4.1 Pre Plate Cleaning/Desmear. Prior to any through hole plating the panel must be cleaned/desmeared. This can be accomplished by 2 basic means. The first method used is plasma desmear. Follow manufacturers guidelines for set up and operation of plasma equipment. The second method is chemical desmear usually achieved by using a sodium or potassium permanganate solution preceded by a solvent solution to enhance the effect of the permanganate. In any desmear operation caution must be taken to reduce the attack on the epoxy due to the lack of fiberglass volume in T-preg dielectric. A guideline to follow is to reduce the cycle time by 50% in any of the desmear operations and measure the effectiveness before proceed. Modifications to the cycle time may be required.

4.2 Hole Metalization Methods. There are many methods to create PTH’s in the DSL panel. We will define the most common processes of hole metalization. T-lam and T-preg materials are compatible with most PTH processes.

4.2.1 Electroless Copper. Low, medium and high deposition baths are compatible with T-lam materials. No specific changes to the cycle are required. Weight gain coupons are recommended to ensure complete coverage and proper deposition rate.

4.2.2 Direct Metalization. T-lam and T-preg is compatible with most direct metalization chemistries. Trial runs to prove coverage and reliability are recommended before production quantities are processed.

4.2.3 Flash Cu Plate. Some manufacturers require a copper flash plate prior to primary image. There are no known problems associated with this process. Follow established electroplating procedures and handle T-lam DSL panel with care as thin laminates can bend and crack easily.

5.0 Primary Imaging

Imaging of the prepared DSL laminate is accomplished under semi clean room conditions utilizing the processes described below.

5.1 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.

5.2 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.

5.3 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.

5.4 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 for temperature, speed and solution makeup.

6.0 Wet Processing

Wet Processing can be one of the most difficult parts of T-lam DSL fabrication. Thin core material can be a challenge to most pcb fabricators. Care and attention in racking, handling and transportation is required.

6.1 Electrolytic Plating. T-lam DSL is compatible with most acid copper and tin or tin/lead plating baths.

6.2 Dry Film Stripping. Follow manufacturers guidelines.

6.3 Etching. T-lam does not react to ammoniacal, cupric chloride or ferric chloride etchants. Follow standard etching procedures for copper weights. Handling procedures for thin core material should be followed.

6.4 Etch Resist 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 and process details.

6.5 Post Etch Cleaning. 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.

6.6 Copper Adhesion Promotion. Prior to lamination to the base metal material the etched DSL panel requires copper adhesion promotion (oxide treatment). Most black, brown or other adhesion promotion chemistries work satisfactory with T-lam materials. Aftr application a bake cycle is recommended to remove any moisture from the DSL panel. A 30 minute bake at 150 C is usually adequate.

7.0 Lamination Cycle

The lamination cycle or bonding operation establishes the mechanical and electrical integrity of the DSL/IMpcb product. Following guidelines can help to ensure uniform panel characteristics, even bond line and good final test results. The press procedure is a starting point and as experience increases you may find better methods and modifications to the procedure.

7.1 Material and Base Metal Preparation. Aluminum base material can be cleaned by using an in line scrubber or deburr machine equipped with 240 – 320 grit bristle brushes. A sufficient water rinse and heated dryer are recommended. All aluminum panels should hold at least 30 sec no water break after surface scrubbing.

Copper base material needs adhesion promotion to allow sufficient surface area for the T-preg to bond properly. Black, brown, gold or other oxides work satisfactorily. Tin and tin/silane treatments also work well.

All panels should be dry and free from contamination prior to lay-up.

7.2 Lay Up Preparation. Follow the procedure as outlined below.

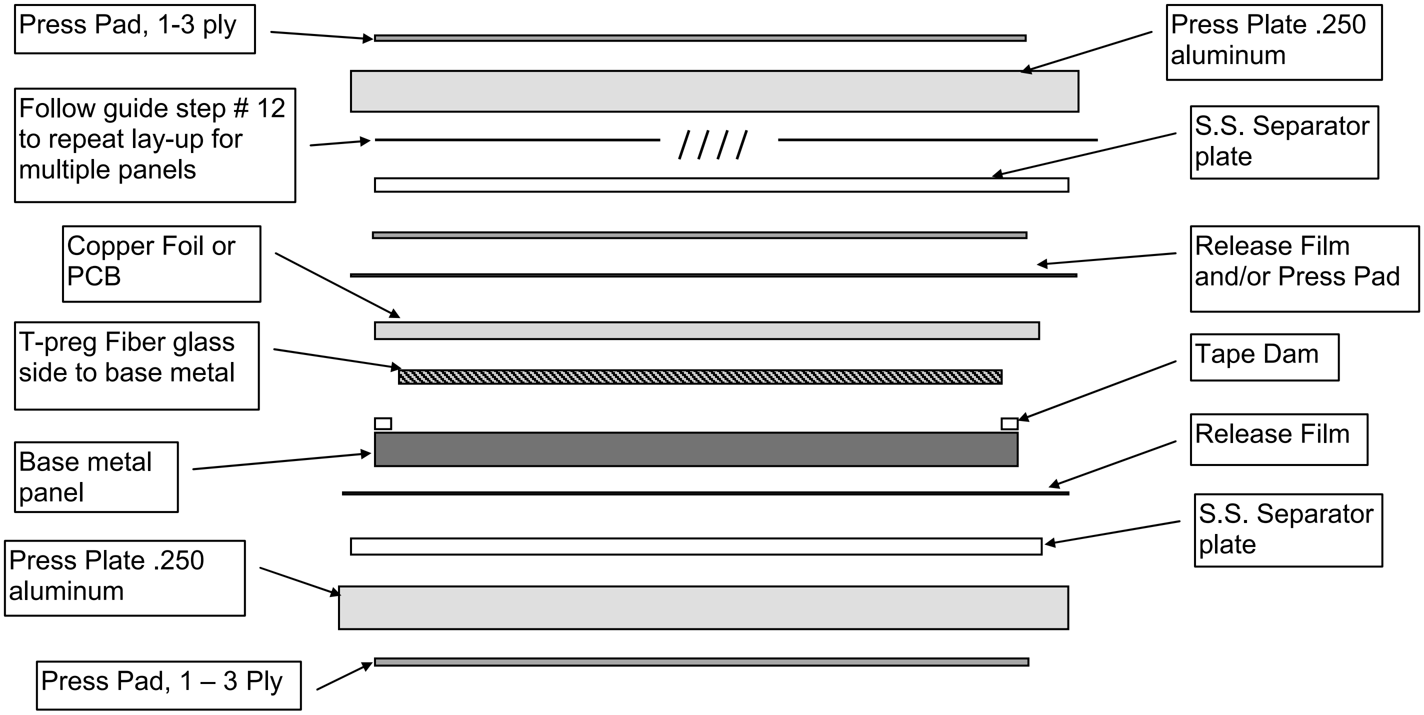
7.2.1 Prepare base metal by either mechanical cleaning and/or chemical cleaning. Clean and dry panels prior to lay-up. Aluminum panels must hold a minimum 30-sec no water break to pass.

7.2.2 Clean and prepare press plates for booking operation. Inspect for dents, pits, resin flash and surface irregularities. Remove plates that cannot be prepared.

7.2.3 Have correct quantities and sizes of T-preg, press pad, release film and copper foil.

7.3 The Lay up sequence from ***bottom to top*** is as follows: (see diagram below)

7.3.1 Press pad or lag material
7.3.2 Press Plate (or Caul Plate) .250” aluminum or steel
7.3.3 Separator Plate (preferably S.S.) .020” - .050” thick
7.3.4 Release Film
7.3.5 Base metal panel scrubbed side facing up
7.3.6 Apply and center T-preg on top of base metal panel (if fiberglass reinforced leave glass down toward base metal)
7.3.7 Tape dam around edge and seal (See T-preg Dam Procedure next page for details)
7.3.8 Place Copper Foil oxide down or PCB circuit side down on top of T-preg. When bonding a PCB use standard inner layer oxide treatment prior to lamination
7.3.9 For foil laminate use Release Film with no press pad, for PCB laminate use conformable release film (Senform or Paco-Via) with a single ply of press pad (PacoPlus) on top
7.3.10 Separator Plate if single panel proceed to step 13 otherwise follow step 12
7.3.11 Repeat steps 5 - 11 up to 25 panels per book
7.3.12 Press Plate (or Caul Plate) .250” aluminum or steel



7.4 T-preg Dam Procedure

7.4.1 Use T-preg material .6" smaller than panel dimensions

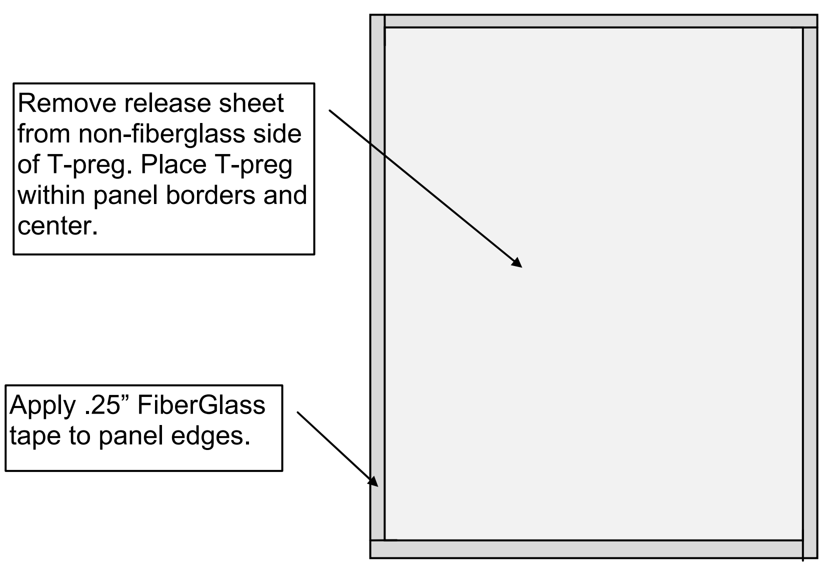
7.4.2 Remove release liner from 1 side of T-preg (if fiberglass 1 side remove fiber glass side)

7.4.3 Place T-preg exposed side on foil or metal panel and center it leaving a consistent even border around the edge of the panel

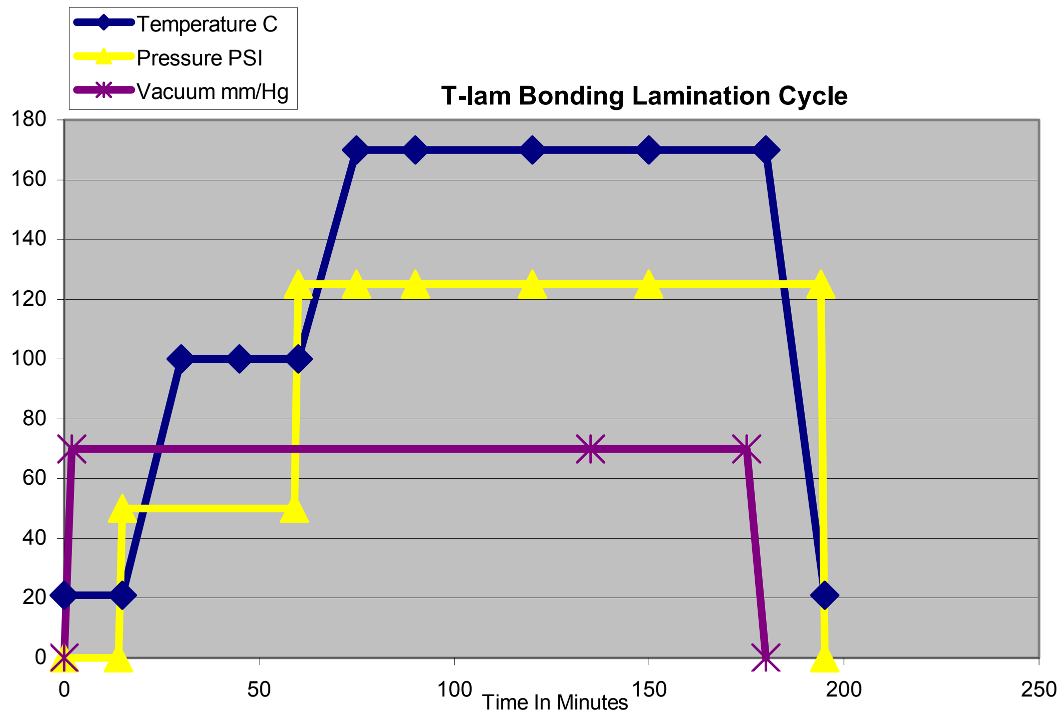
7.4.4 Apply .25" glass reinforced tape (Thermagon Pt # A10326) to the borders, aligning it carefully without any gaps or overlap. Secure tape to panel by rolling or pressing making sure complete adhesion occurs

7.4.5 Carefully remove 2nd release liner from top side of T-preg

7.4.6 Panel is now ready to continue booking.



7.5 Press Procedure. The following cycle is a guideline for bonding DSL and other pcb's to base metal substrates with T-preg material.



1. Bring Vacuum to 70 mm/Hg (or 29"/Hg)
2. Hold at room temp 21 C for 30 minutes
3. Add pressure to 50 PSI and increase temp to 100 C rate of rise 5 C/minute
4. Hold at 50 PSI and 100 C for 30 minutes.
5. Increase pressure to 125 PSI and increase Temp to 170 C rate of rise 5 C/minute
6. Hold at Temperature and pressure for 90 minutes minimum
7. Reduce temperature to 21 C rate of decrease 7 - 9 C/minute
8. Remove pressure when temperature ~< 30 C

7.6 De-Booking, epoxy removal and Post Press Bake. Exercise care when de-booking the panels and inspect for excessive epoxy squeeze out in the PTH areas of the panel. Epoxy removal from the surface of the panel can be achieved through the use of mechanical, chemical or plasma technology. Experimentation using the process is recommended prior to production use. Plasma and mechanical removal has proven to be the most consistent and non damaging to the base metal substrate.

7.7 Panels can be post press baked for 1 – 1 ½ hours at 175 C to ensure total cure and to stress relieve the material. This is an optional process and is not a mandatory step.

8.0 Post Lamination Drilling/Punch and Second Step Drilling

After lamination and epoxy squeeze out removal the panels are ready for registration tooling and second step drilling. These operations involve machining of a composite material with a solid metal base that prove challenging to PCB fabricators. Exercise extreme caution when performing these operations, as attention is required to succeed in proper location and quality of tooling holes.

8.1 Post Lamination Registration. After lamination tooling holes will be required to align solder mask and any secondary machining operations i.e.… NPTH's, v-score tooling and other fabrication steps.

8.2 Spot Face Drill. Has been the most accepted means of locating and adding registration hole tooling after lamination. Due to the base metal substrate drilling is usually less destructive than post lamination punching. Use feed rates 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. Drill feeds and speeds should match that of solid metal removal settings. Low chip loading and slow in feed rates are recommended.

8.3 Post Lamination Punch. In the event that spot face drilling is unavailable, post lamination punching can be used to locate and add registration holes. Care must be taken as solid metal substrate punches with more force required than typical FR-4 multilayer panels. Refer to equipment manufacturers suggestion prior to attempting this operation.

8.4 Second Step Drilling or Pierce Punching.

8.4.1 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.

8.4.2 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.

9.0 Solder Mask Preparation and Application

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 Methods used. Prior to any solder mask material a clean surface is required. Mechanical scrubbing using a 320-grit bristle brush, pumice slurry or chemical cleaning will work sufficiently.

9.1 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.

9.2 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.

9.3 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.

9.4 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.

10.0 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.

10.1HASL. 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.

10.2OSP. 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.

10.3 Immersion or Electroless Tin. 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.

10.4 Ni/Au. 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.

11.0 Fabrication

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

11.1 Rout. 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" – 40"/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.

V-Score. 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.

11.2 Punch. 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.

12.0 Final 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.

12.1 Electrical Test. Standard electrical test fixtures will work for continuity and open testing on T-lam pcb's. Net list testing to ensure 100% electrical integrity is recommended. Many universal grid and flying probe testers work with T-lam circuit boards. Contact test equipment manufacturer for any specific details.

12.2 Hipot Testing. 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.

A document on fixture construction is available upon request.

13.0 Packaging

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

13.1 Packing. 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.