PRINTED CIRCUIT BOARDS

PROCESS CAPABILITIES
AND
DESIGN FOR MANUFACTURE GUIDELINES

VERSION 3.75
RELEASED 07/05/2024.

TABLE OF CONTENTS

NOTE: This document serves a dual purpose.

The first section outlines Lintek's manufacturing and process capabilities, including standard board types and typical manufacturing tolerances. It is important to highlight that while we specialise in producing boards for microwave and high frequency applications, we have a wide range of production capabilities that suit other industry sectors.

The remainder of the document provides Design For Manufacture “DFM” guidelines, suggesting some best practice design rules for simple manufacture of PCBs. Designing your board in accordance with these guidelines will ensure a smooth transition to production at Lintek.

In many cases, we are also able to tighten our process tolerances if you have specific requirements. Our sales team will be only too happy to discuss your specific requirements and even develop a tailor-made solution for your application. Please contact **sales@lintek.com.au** for any sales enquiries or for further information on Lintek's process capabilities.

INTRODUCTION

Lintek Pty Ltd is an Australian company that has specialised in producing microwave and high frequency printed circuit boards for over 30 years. Our unique vacuum copper metallisation process reliably produces clean, consistent copper traces with unparalleled accuracy and excellent impedance characteristics. Our innovative, simplistic, low cost and low risk process produces world class products for a wide range of commercial and defence applications.

Lintek is built on a tradition of innovation and commitment to customers to deliver the best product possible for each application with consideration to cost versus performance at every stage. Our motto "Surely there's a better way" is fundamental to our continuous improvement strategy utilising the principles of "Six Sigma" and "Lean" process methodologies. To support our endeavour Lintek is heavily invested in quality management systems and is ISO 9001:2015 + AS9100D and DISP certified.

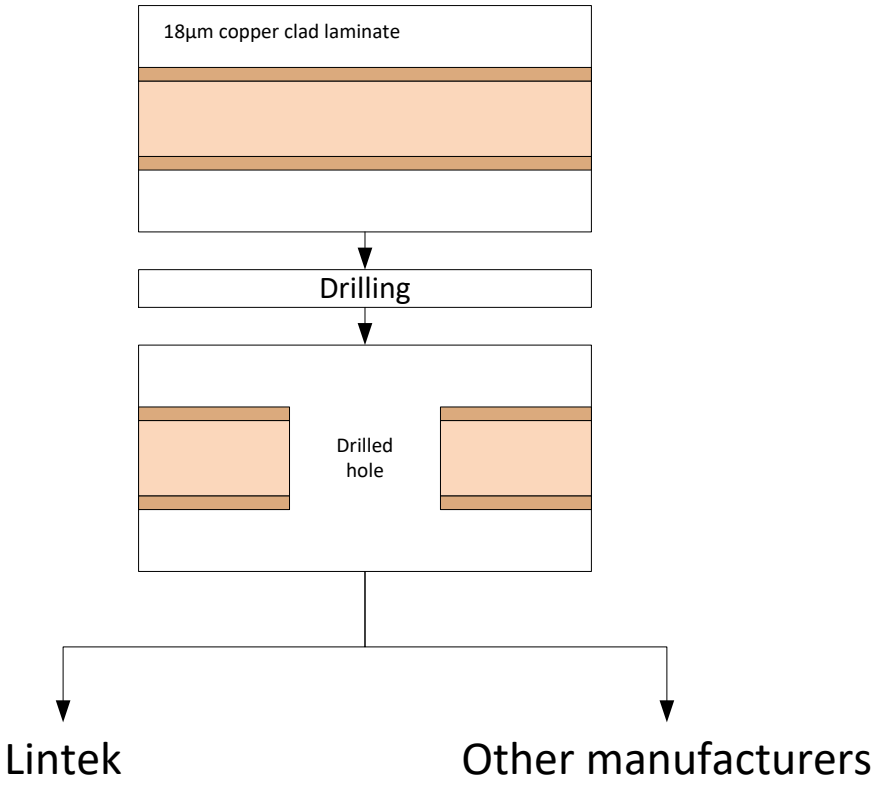
THE LINTEK DIFFERENCE

Lintek has established a strong presence in the microwave and high frequency market by producing boards with highly accurate features and excellent impedance characteristics. Our customers like the fact that they don't have to tune our boards as we deliver copper features that match their designs and simulations.

Lintek can offer such excellent features due to our unique manufacturing process, vacuum metallisation. By vacuum depositing a 2µm seed layer on bare panels instead of starting with 18µm foil, the final panel etch is reduced to a 2µm microetch. This eliminates undercut and produces straight, accurate side walls, critical to RF performance.

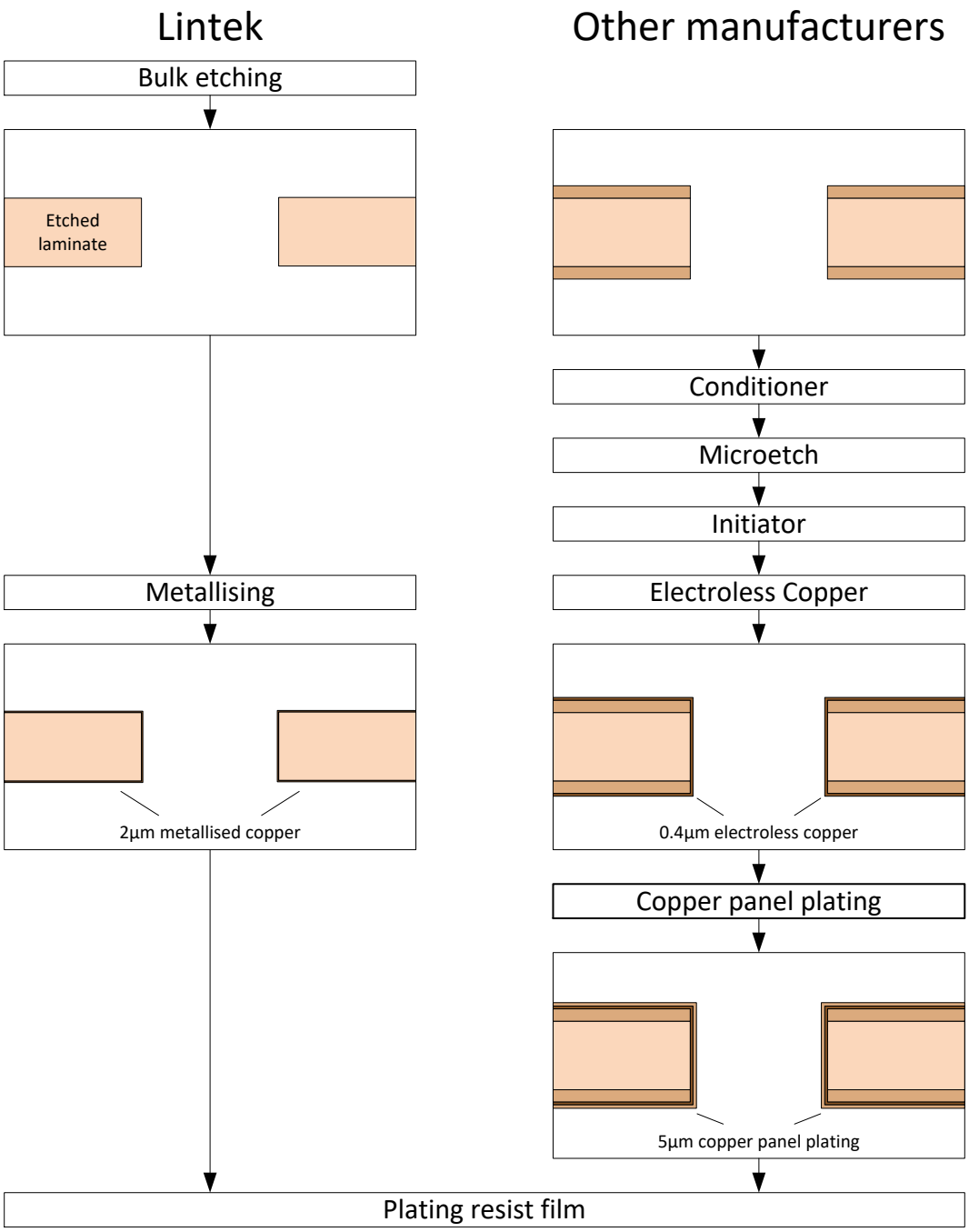
Our method requires fewer process steps, uses less energy and chemicals than traditional manufacturing methods, making Lintek's process significantly more environmentally friendly.

Our manufacturing process starts like other manufacturers. We drill holes in a copper clad laminate, usually starting with 18µm of copper on the panel surface. Leaving copper on the surface before drilling protects the dielectric surface from damage and acts as a guide to keep drill bits from slipping on the etched surface that forms the unique part of Lintek's process.

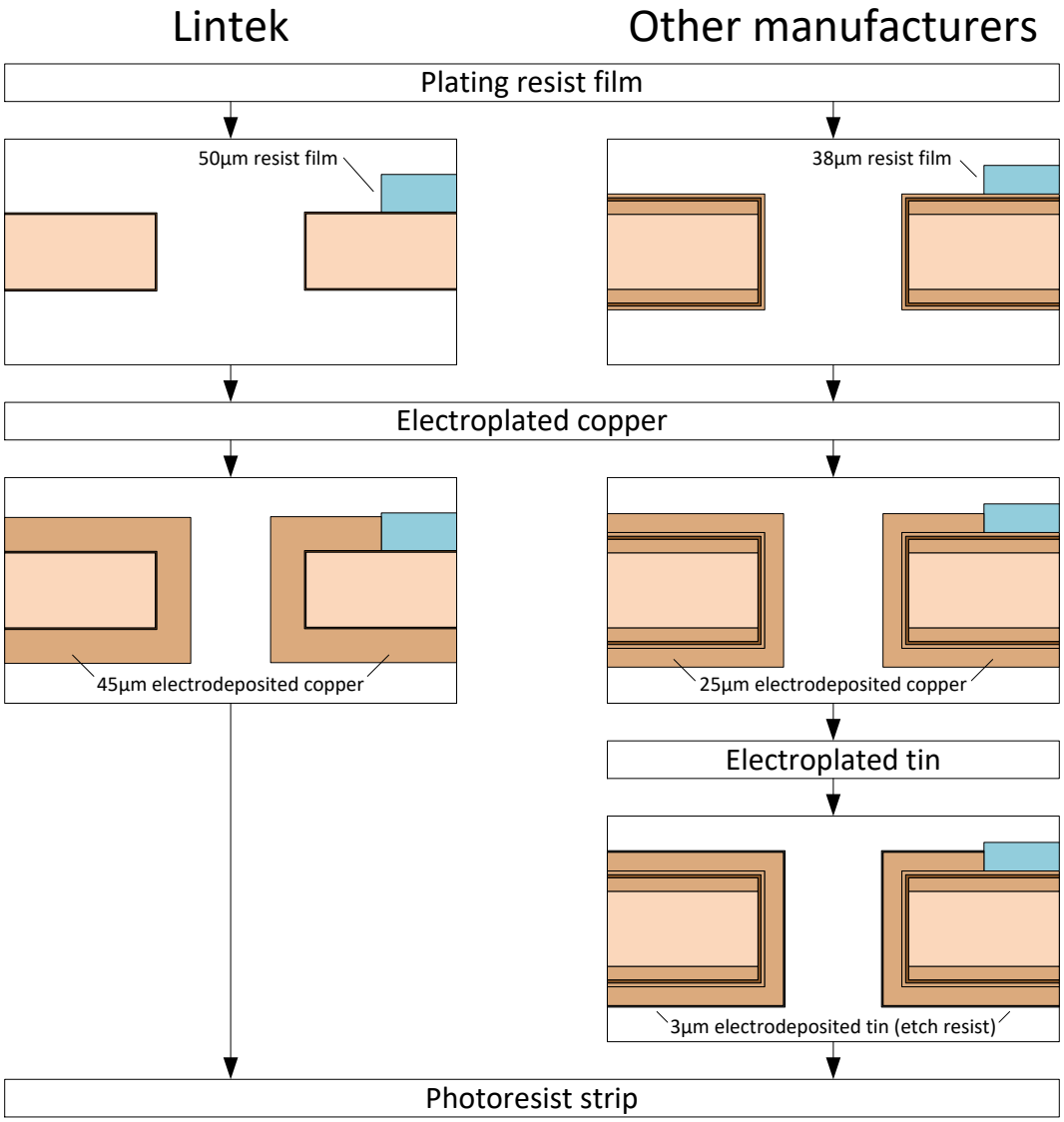


At this point, most manufacturers apply an electroless copper seed layer to enable hole plating. This is the first of several layers of copper that they will add before stripping back a large portion at the end of the process. Electroless copper plating requires surface preparation and often involves aggressive chemicals.

Lintek, by contrast, strips away all copper on the panel and applies a 2µm seed layer using our unique vacuum deposition process. Since fewer chemical processes are required, this is much more environmentally friendly and leads to a reduction in waste and chemical by-products.

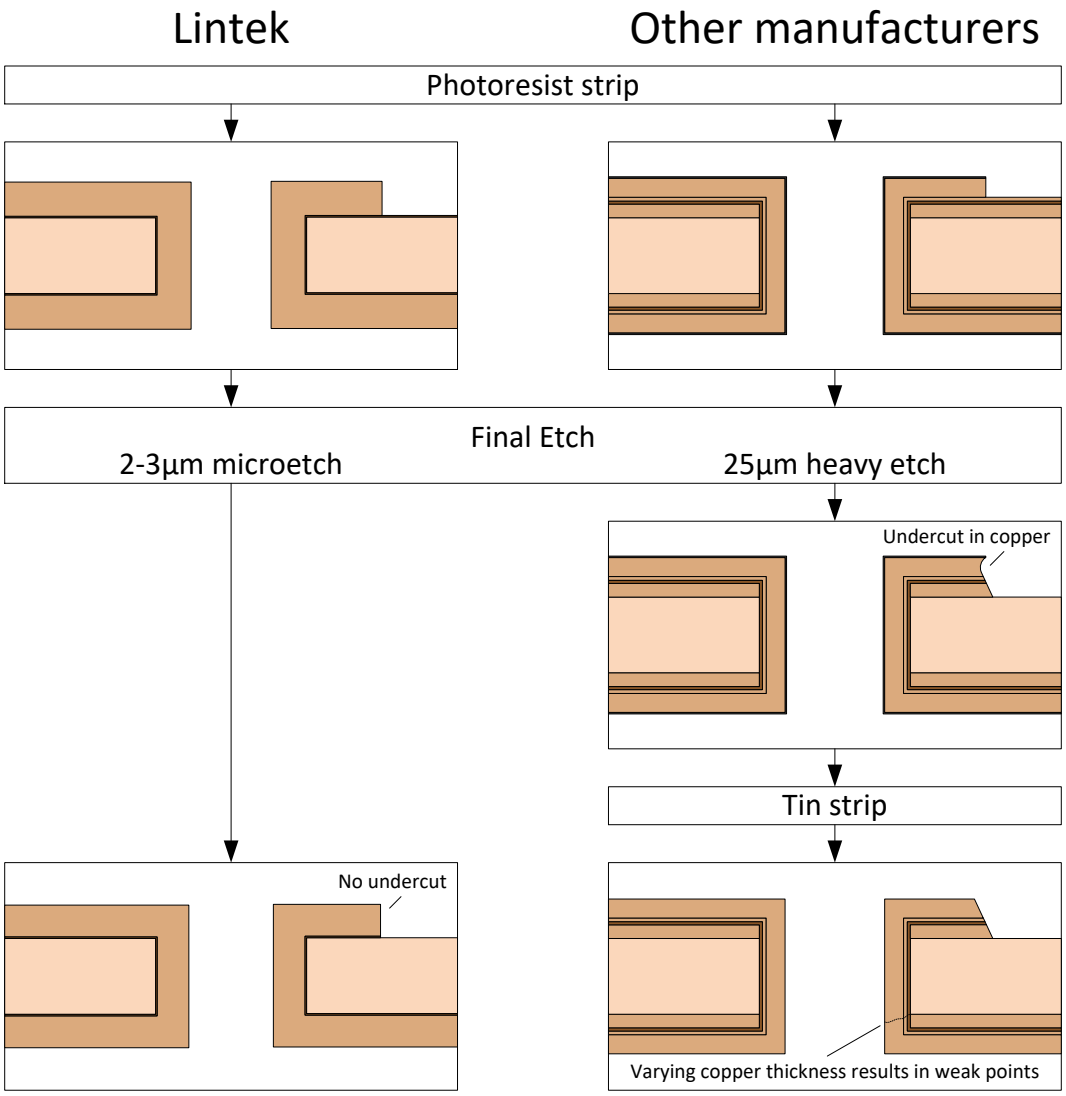


With this seed layer applied, we can then pattern plate the features on the panel with a standard copper electroplating process. After applying a photoimageable plate resist layer, Lintek only requires one process that electroplates 45µm of copper onto our seed layer. This single pass copper plating process will plate the surface features and through hole connections in once seamless operation producing significantly thicker Cu sidewalls in the Plated Through Holes meeting IPC Class III minimum each time, we produce a board. Other manufacturers must also electroplate a Tin etch resist layer over copper in preparation for final etchback.



When the photoresist is removed, the exposed copper can be etched back to expose the substrate. Lintek only performs a small “micro-etch” at this point, removing the 2µm of copper to strip away the metallised seed layer. Stripping such a small amount of copper means that no etch resist is required. It also means that electroplated finishes such as Nickel and Gold can be applied for high reliability wire-bonding with minimal undercut at the Ni/Cu interface typically seen with other processes.

By contrast, other manufacturers perform a heavy etch. The removal of 25µm of copper requires a much longer exposure to the etchant and results in substantial sidewall etching and variability in trace widths. An additional process is also required to strip the layer of tin from the finished board.



In the standard process, the variation in copper thickness between the barrel plating and the surface can produce weakness in the corners of plated through holes. Since Lintek’s process maintains uniform plating on all surfaces of the panel, the plated through holes are more reliable under a range of conditions.

Lintek’s vapor deposition process eliminates many chemical compounds and process steps used in the manufacture of printed circuit boards, including palladium chloride, formaldehyde, complexed coppers, reducing agents, stannous chloride, sodium etch, tin plating and tin stripping.

A full comparison between our method and a standard manufacturing process is shown in *Appendix 1* – Process Comparison.

PCB TYPES

We are able to produce boards of the following types:

- Single sided
- Double sided
- Edge plated
- Cavity multilayers
- Multilayers with up to 16 layers, including mixed dielectrics
- HDI multilayers with laser drilled micro vias, buried, blind and stacked vias
- Metal cores “coining” embedded in multilayers with isolated feed through vias
- Metal backed PTFE including blind vias

Lintek has experience working with a wide range of materials. Available substrates include:

- Polyimide
- FR4
- Ceramic thermoset
- PTFE and ceramic loaded PTFE
- CNC machined metal backed carriers

Lintek maintains a stock of standard and specialist materials. We are also able to work with customer supplied laminates.

PANEL SIZES

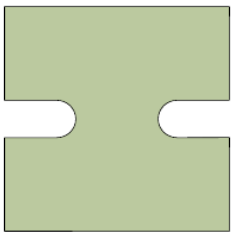
Board Type	Maximum Panel Size		Maximum Usable Area		Maximum Thickness (mm)
	in	mm	in	mm	
Single Sided	18×24	457×610	17×23	432×584	3.2
Single Sided (Long)#	24×72	610×1829	23×71	584×1803	3.2
Double Sided	18×24	457×610	17×23	432×584	3.2
Double Sided (Long)#	24×72	610×1829	23×71	584×1803	3.2
Double Sided with Plated Through Holes	18×24	457×610	17×23	432×584	3.2
Multilayer	18×24	457×610	17×23	432×584	6*
Metal Backed	16×12	406×305	15×11	381×280	6

* Custom built multilayers using thicker laminates are available on request.

\# Long boards up to 1800mm in length can be produced without plated through holes, solder mask or legend.

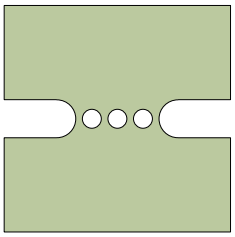
BOARD LAYOUT

When positioning boards on the usable area of a panel, we offer the following options:



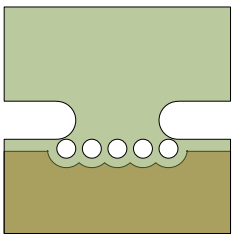
ROUTED TAB

Route along the outline of the board, leaving tabs to hold the boards on the panel until manually separated.



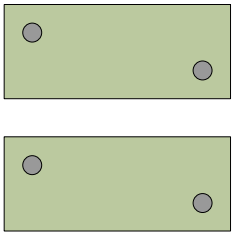
DRILLED TAB (STANDARD)

Similar to Routed Tabs but with a series of holes drilled in the tab. Provides a cleaner break with less clean-up required.



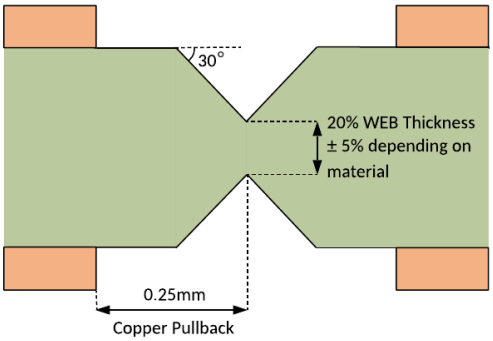
DRILLED TAB (BLIND SNAPOFF)

A variety of drilled tab that ensures external dimensional tolerances are maintained without further finishing. Requires additional clearances between drilled holes and copper features.



PINNED ROUTE

Pins hold the boards in position while the outline is routed. Provides the cleanest finish but requires 2 positioning pins on every board.



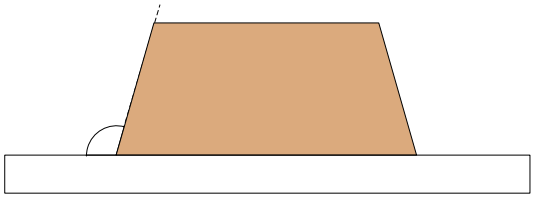
VEE GROOVE

V-Groove scoring, or V-Scoring is a “Vee” cut into both sides of a printed circuit board to enable easy separation of parts after assembly.

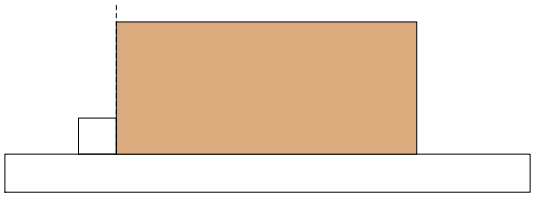
PROCESS TOLERANCES

ETCHING

Due to Lintek's unique metallising process, we are able to produce conductors with an etch factor of zero. By starting with a thin copper seed layer, the amount of copper removed in the final etch is reduced significantly. This method decreases undercut and produces cleaner features with straight sidewalls.



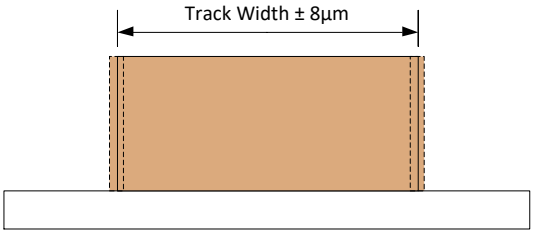
Typical profile of etched conductor

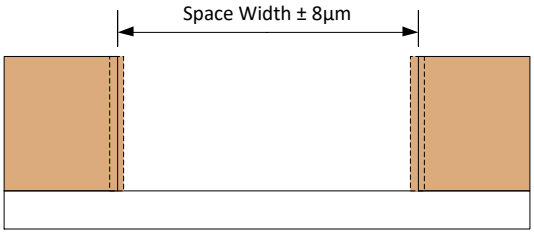


Straight walls produced by Lintek process

The standard tolerance on the width of electroplated tracks and spaces is ±0.0125mm.

Using our special “additive” process the tolerance on the width of electroplated tracks and spaces can be reduced to ±0.008mm.*





*The etch tolerance can vary slightly with different materials and surface finish requirements. Our technical team can assist with answers specific to your material and finish requirements.

COPPER THICKNESS

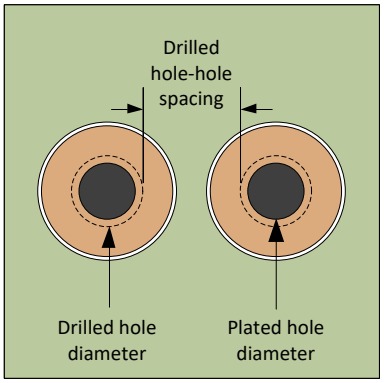
Lintek prefers to have a minimum of one-ounce electroplated copper for all Plated Through Hole applications. The copper plating has a finished thickness of 35^{+10}_{-0}microns. Very accurate track and space with minimal sidewall undercut is also achievable with copper traces up to 100 microns thick. Heavy copper electroplating up to 500 microns is available on request.

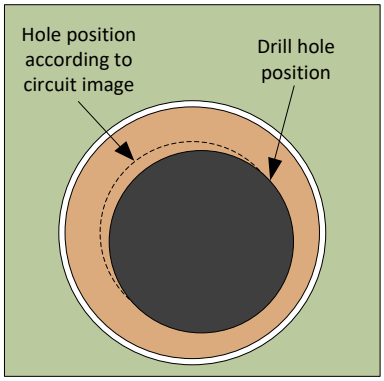
MECHANICAL DRILLING

Lintek's CNC drill machines are capable of maintaining high accuracy and repeatability. Due to material and process restrictions, we suggest that the following tolerances and restrictions be taken into account during board design.

Characteristic	Value
Hole repeatability	±20 microns
Minimum drilled hole-hole spacing (edge to edge and is dependent on laminate thickness and drill bit diameter)	200 microns
Controlled depth drilling precision	±50 microns
Alignment to circuit image	±100 microns
Minimum hole diameter (dependent on laminate)	0.15mm
Maximum drill diameter*	6.5mm

* Holes larger than the maximum drill diameter will be routed rather than drilled.





Finished hole size tolerances are as agreed between user and supplier. If no tolerance is specified, Lintek maintains default tolerances as shown on the table below.

For material up to 1.6mm thick:

Finished Hole Diameter (mm)	Tolerance – Non-plated Through Holes (mm)	Tolerance – Plated Through Holes (mm)
Less than 0.8	±0.05	±0.08
0.8 to 1.6	±0.08	±0.10
1.6 to 5	±0.10	±0.15

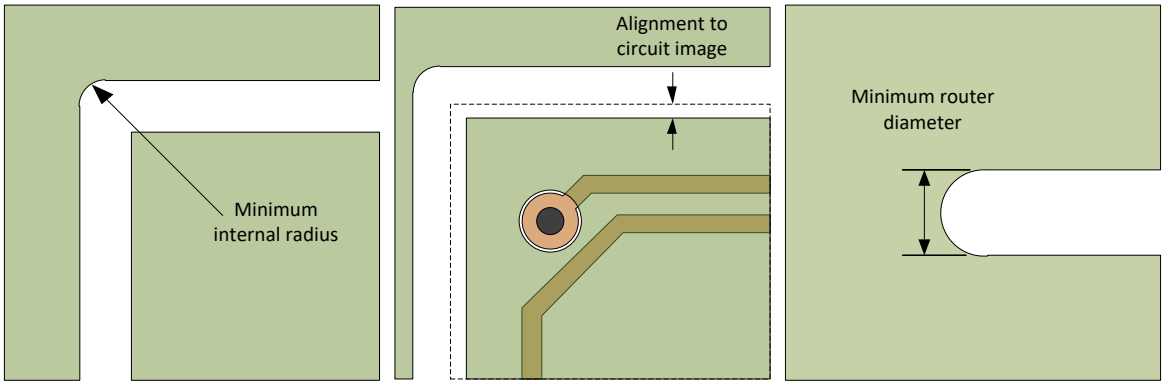
A further ±0.03 is added for substrates thicker than 1.6mm.

ROUTING

Lintek's CNC routers also maintain excellent process tolerances.

Characteristic	Value
Profile tolerance	±100 microns
Minimum internal radius of routed area	0.20mm
Standard router diameter	2.4mm
Minimum allowable router diameter*	0.4mm
Alignment to circuit image	±75 microns

* Dependant on material type and thickness. Use of small diameter routers may increase costs due to wear and tool breakage.



MILLING

CNC milling of solid metal backing can provide higher precision than standard drilling and routing in some materials.

Characteristic	Value
Hole position accuracy or repeatability	±20 microns
Minimum drilled hole edge-edge spacing (dependent on laminate)	100 microns
Controlled depth milling	±50 microns
Profile tolerance	±50 microns
Minimum internal radius of routed area	0.25mm
Standard Milled surface roughness (R_a)	2.5 microns
Largest panel size	305mm x 406mm

LASER DRILLING/MACHINING

Laser drilling and profiling allows accurate machining not achievable when using mechanical drilling or profiling equipment. Lintek has developed laser machining capabilities for a wide range of exotic PTFE and standard FR4 substrates. For further information please contact sales@lintek.com.au to discuss compatible materials and options/limitations for laser drilling and machining.

Characteristic	Value
Hole position accuracy or repeatability	±10 microns
Hole diameter accuracy	±12.5 microns
Hole drilling diameter range	25-500 microns
Minimum beam diameter	25 microns
Minimum drilled hole-hole edge spacing (laminate dependent)	50 microns
Profiling accuracy	±25 microns
Minimum internal radius of routed area (= min beam radius)	12.5 microns
Top/bottom Cu layer pullback requirement for laser profiling	50 microns
Copper filled blind microvias (1:1 aspect ratio required)	50-150 microns
Controlled depth drilling accuracy to buried copper layer	±5 microns
Deepest hole	500 microns

DESIGN FOR MANUFACTURE GUIDELINES

These design rules provide a guide that will allow clients to produce a board that can be manufactured by Lintek with minimal modification of the design. Please note that designing to special tolerances will attract a premium.

MULTILAYER BOARD LAYUP GUIDE

Lintek is capable of producing multilayer boards with 4 or more layers. We have considerable experience implementing mixed dielectrics within a multilayer stack to suit the dielectric requirements of high frequency and microwave circuits.

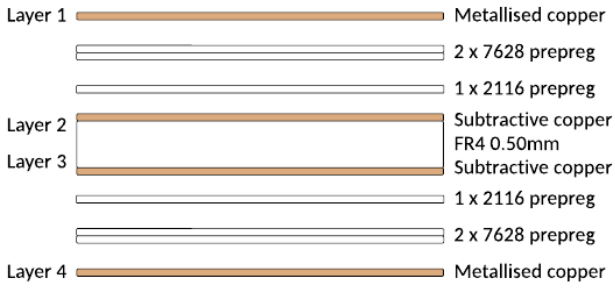
We use our proprietary metallising technology to produce excellent impedance characteristics on the outer layers of a board, and use low-cost subtractive layers for internal power and ground planes. We are able to include precisely plated inner Cu layers for controlled impedance features if required.

Multilayer stacks should be designed symmetrically from the centre. Asymmetrical substrate choice will often produce bowing or twisting in the finished board.

Lintek has standard FR4 layups for 4, 6, and 8 layer boards using NP-140B materials (see following table). Choosing a standard layup for your board design will accelerate release to manufacture as these layup configurations have been regularly used and thoroughly tested.

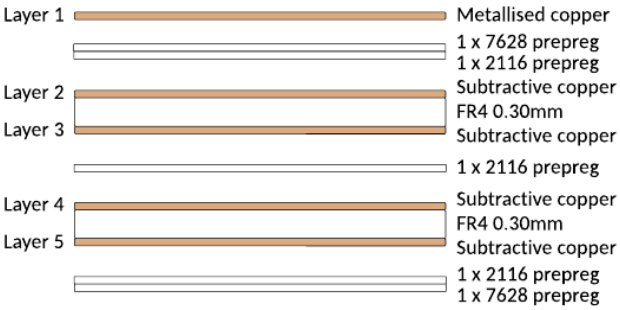
If you are considering a more complex design or a multilayer that includes mixed dielectrics, you are welcome to contact us during the design phase of your board and we will assist in ensuring the manufacturability of your design.

4 LAYER BOARD



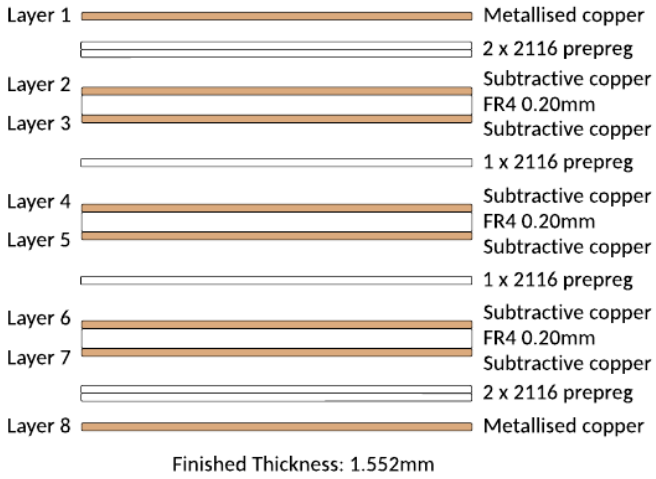
Finished Thickness: 1.620mm

6 LAYER BOARD



Finished Thickness: 1.527mm

8 LAYER BOARD

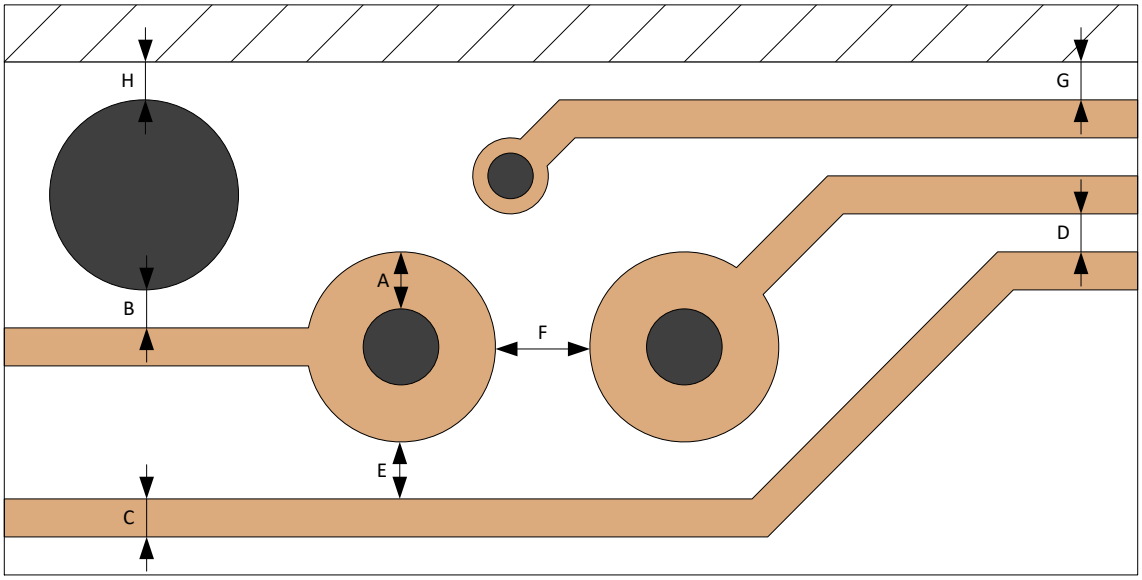


The properties of our *commonly used* FR4 and RF prepregs are shown below to aid impedance calculations. Detailed Dk/Df charts are available online and we can assist if you require more information.

Material	(Tg) Glass Transition Temperature °C	Prepreg Glass Style	Thickness (mm)	Testing Frequency	D_k	D_f
NP-140B	**140±5**	**106**	**0.053**	**1MHz**	**4.2-4.4**	**0.015-0.020**
NP-140B	140±5	1080	0.065	1MHz	4.2-4.4	0.015-0.020
NP-140B	**140±5**	**2116**	**0.105**	**1MHz**	**4.2-4.4**	**0.015-0.020**
NP-140B	140±5	7628	0.180	1MHz	4.2.4.4	0.015-0.020
PCLFR370HR	**180**	**106**	**0.058**	**100MHz**	**4.24**	**0.015**
PCLFR370HR	180	1080	0.076	100MHz	4.24	0.015
PCLFR370HR	**180**	**2113**	**0.102**	**100MHz**	**4.24**	**0.015**
PCLFR370HR	180	2116	0.122	100MHz	4.24	0.015
PCLFR370HR	**180**	**7628**	**0.185**	**100MHz**	**4.24**	**0.015**
FR406NLowflow	170	1080	0.069	100MHz	4.00	0.013
RO4450F	**280**	**2113**	**0.101**	**10GHz**	**3.52**	**0.004**
MW9350	200	2113	0.101	10GHz	3.50	0.004
AR6700	***N/A**	**Bondfilm**	**0.038**	**10GHz**	**2.35**	**0.0025**

*Note: AR6700 is a low melting point thermoplastic bonding film with a Crystalline melting Point of 184°C. The film will re-melt at temperatures above this Point and it has a Continuous Use Temperature of 176°C – Care must be taken during assembly to avoid core delamination.

EXTERNAL LAYERS



MINIMUM DIMENSIONS – SIGNAL LAYERS

Dimension	Description	Standard		Special	
		mm	thou	mm	thou
A	Hole to pad (minimum annular ring)*	0.190	7.5	0.1397	5.5
B	Track to non-plated hole	0.254	10	0.1524	6
C	Track width	0.152	6	0.0508	2
D	Track to track spacing	0.152	6	0.0508	2
E	Track to pad spacing	0.152	6	0.0508	2
F	Pad to pad spacing	0.152	6	0.0508	2
G	Track to board edge	0.254	10	0.0762	3
H	Non-plated hole to board edge	0.152	6	-	-
I	Plated through hole diameter#	0.304	12	0.1016	4

* Specification of the annular ring is based on finished hole diameter after electroplating PTH barrel. The minimum annular ring around a drilled hole (not finished) is the drill diameter + 0.200mm copper pad. For example, a 0.3mm drill will require a 0.5mm pad before copper plating the via. Based on 35 microns of copper plating the finished hole diameter after plating would be 0.230mm (0.3-0.070mm (0.035mm x 2))

\# Diameter may be limited by material thickness. See *Pth Aspect Ratio*.

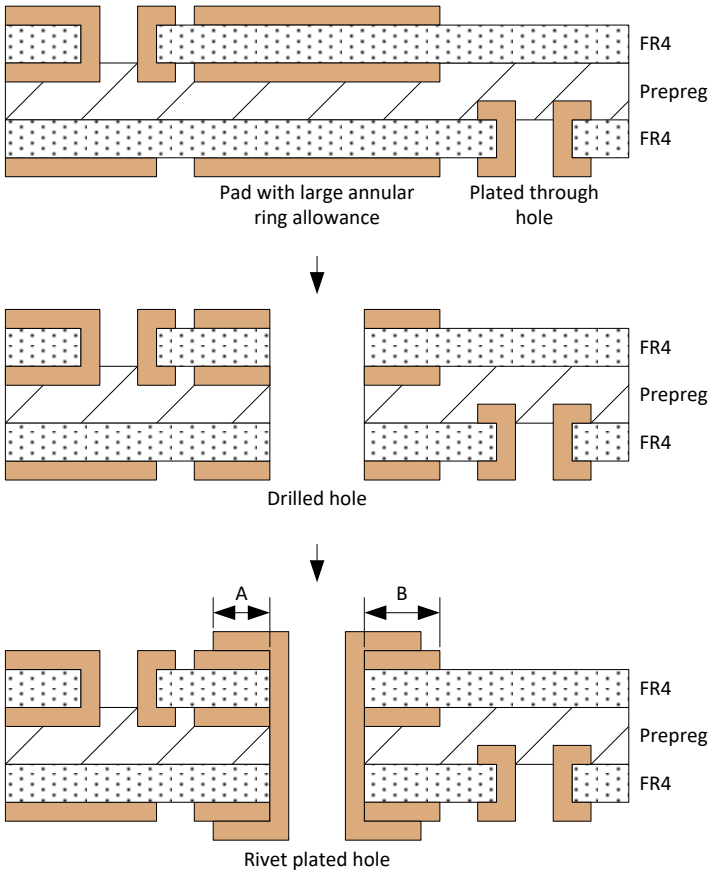
RIVET PLATING (COPPER WRAP)

Depending on Multilayer board material and stackup requirements, Rivet plating may be required to ensure stability and manufacturability.

Rivet plating provides a plated through hole connecting an inner layer to external copper features or connecting external features on opposite sides of the panel. Since they are typically drilled post-lamination, rivet plated holes pass through the entire circuit board. The additional copper plating provides excellent mechanical stability and improves the reliability of multilayers on some materials.

Pads that will be used for rivet plating require a larger than standard annular ring to ensure alignment of the rivet plated hole to the original copper pad.

The diagram below shows the process of rivet plating and a typical cross section of the finished product.



MINIMUM DIMENSIONS – RIVET PLATING

Dimension	Description	Standard		Special	
		mm	thou	mm	thou
A	Minimum annular ring around rivet plated hole	0.127	5	0.076	3
B	Minimum annular ring on pads connected to rivet plate	0.254	10	0.152	6

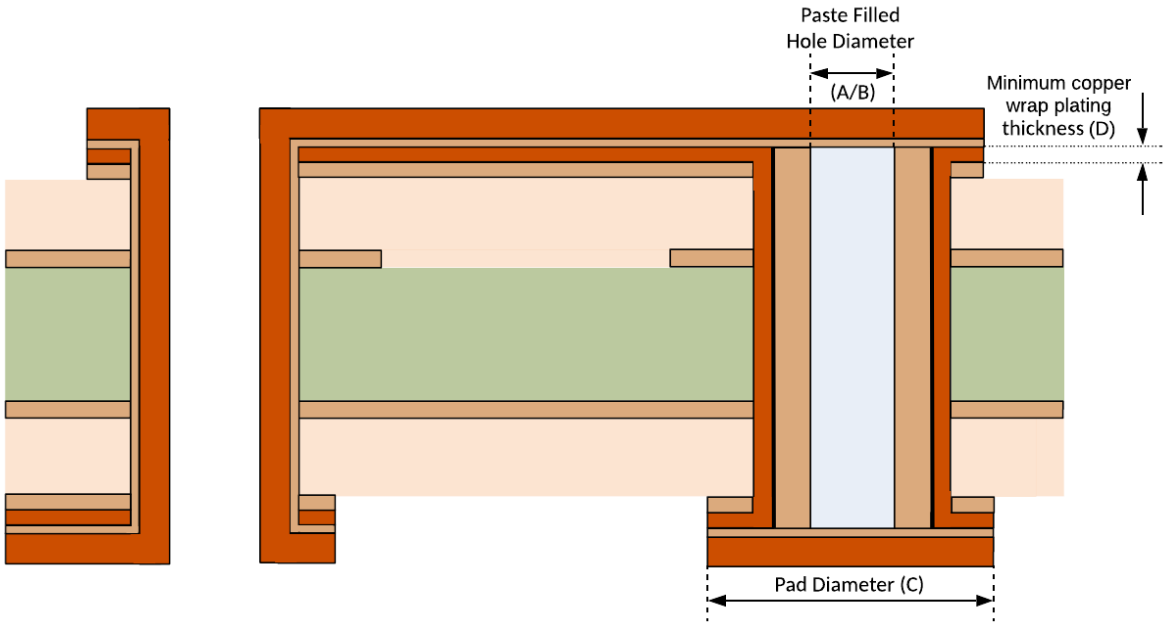
FILLED AND COPPER CAPPED PLATED THROUGH HOLES

Modern High-Density Interconnect (HDI) designs often require vias to be paste filled and copper capped to allow components or devices such as Ball Grid Array's (BGA's) to be mounted directly over interconnection vias on a Printed circuit Board (PCB).

The copper plated vias are filled with a stable non-conductive paste (*such as* Taiyo THP-100 DXP) with a Co-efficient of Thermal Expansion (CTE) closely matched to copper using specialised vacuum filling equipment before being baked and planarized. The copper plated pattern layer and pads are electroplated over the top of the paste filled vias and the subsequent printed circuit pattern begins to take shape.

This process simplifies, PCB layout, Component assembly, and allows PCB design engineers to increase component density by mounting devices directly over the interconnect holes which is not possible if the holes remain open.

The diagram below shows the through hole Via in Pad process and a typical cross section of the finished product.

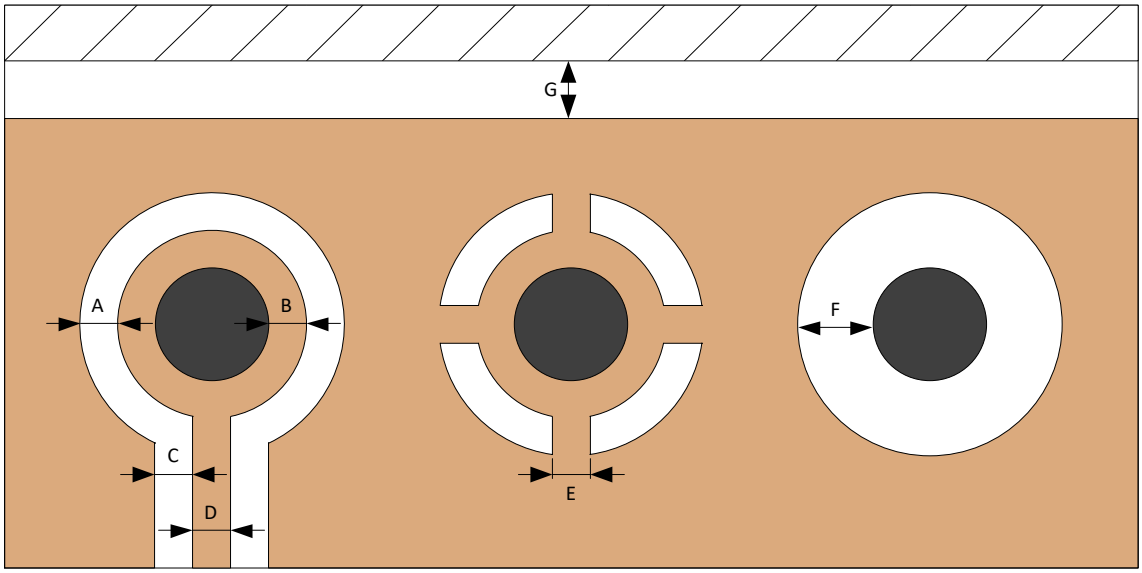


FILLED AND CAPPED VIA DIMENSIONS

Dimension	Description	Standard		Special	
		mm	thou	mm	thou
A	Minimum diameter of the paste filled holes*	0.20	8	0.10	4
B	Maximum diameter of the paste filled holes *	1.60	60	2.4	94
C	Minimum pad diameter for paste filled holes	0.60	24	0.30	12
D	Copper wrap plating thickness (AABUS*)	0.012	0.5	0.006*	0.25

* Min diameter is dependent on material and board thickness. Max diameter equals the board thickness.

INTERNAL POWER AND GROUND PLANES

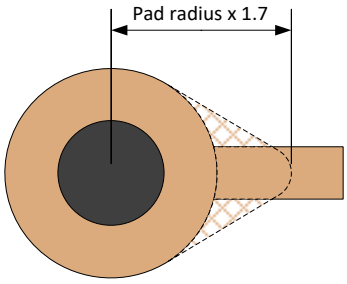


MINIMUM DIMENSIONS – POWER AND GROUND PLANES

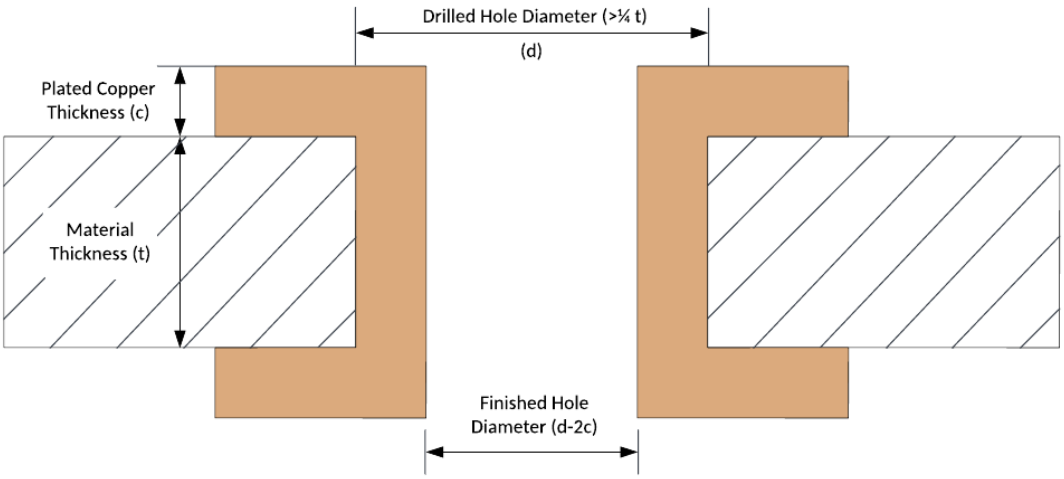
Dimension	Description	Standard		Special	
		mm	thou	mm	thou
A	Plane to pad	0.2032	8	0.1016	4
B	Hole to pad (minimum annular ring)	0.2032	8	0.1016	4
C	Plane to track	0.2032	8	0.1016	4
D	Track width	0.2032	8	0.1016	4
E	Thermal break width	0.3048	12	0.1016	4
F	Plane to hole	0.2540	10	0.1524	6
G	Plane to board edge	0.5080	20	0.1016	4

NOTE - All dimensions in the table above are based on the drill size which is a minimum of 0.10mm larger than the finished hole size.

We encourage designers to add teardrops or signal flares to pads as recommended by IPC 2221 (9.1). Teardrops increase the reliability of the pad-trace junction and can increase yield in multilayer panels. Lintek recommends using a teardrop factor of 1.7, as shown below.



PTH ASPECT RATIO



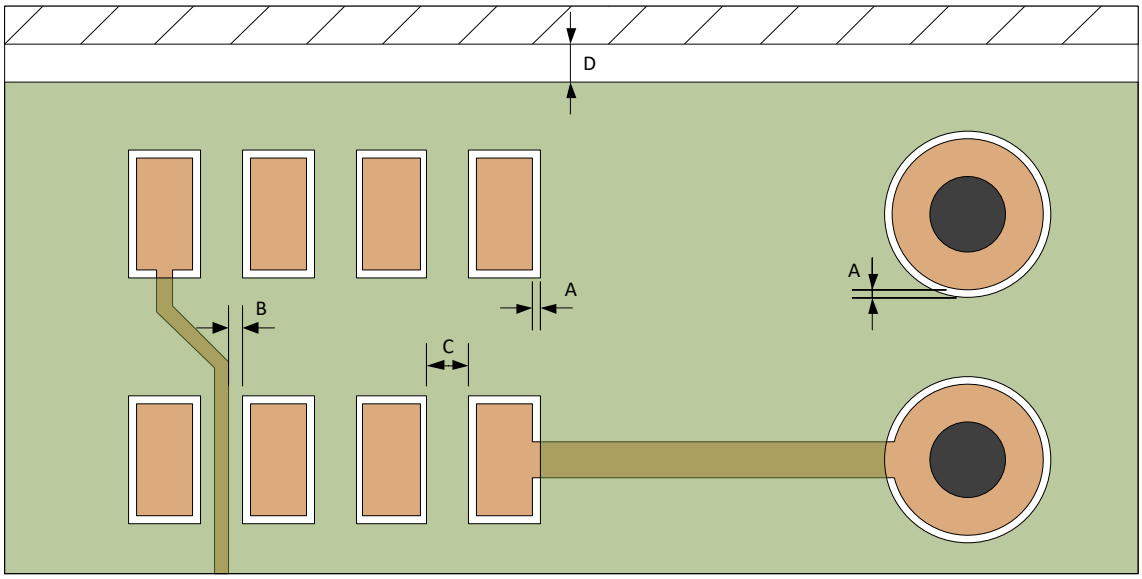
Using Lintek's vacuum copper deposition and additive technology, Lintek is able to produce plated vias with a standard finished hole aspect ratio of 5.3:1. Using our standard additive process, the drilling diameter of any plated through hole must be greater than ¼ the thickness of the layer/s it passes through.

For instance, on a 1.60mm thick substrate, the minimum drilling diameter is 0.4mm. This will provide a finished hole diameter of 0.33mm, assuming one ounce (0.035mm) of copper plating thickness inside the hole barrel.

Lintek has also incorporated other methods to create PTH in high aspect ratio designs to allow for high layer count multilayer PCB designs typically required for HDI production.

For applications where an aspect ratio of greater than 5.3:1 is required, please contact the team at sales@lintek.com.au to discuss other options.

SOLDER MASK



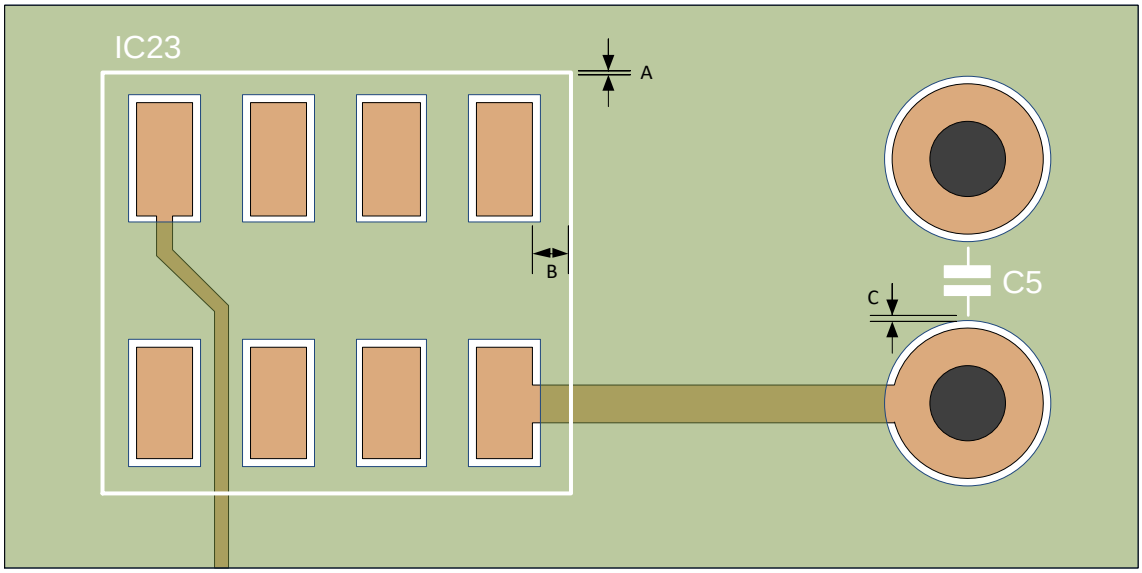
MINIMUM DIMENSIONS – SOLDER MASK

Dimension	Description	Standard	
		mm	thou
A	Mask to pad	0.1016	4
B	Mask to track	0.1016	4
C	Mask width	0.1524	6
D	Mask to board edge*	0.254	10

* Solder mask can be applied up to the edge of a board, but chipping may occur during routing. The given spacing is recommended to ensure clean edges.

Colours available as standard: green, white. On request: Blue, black, red.

LEGEND



MINIMUM DIMENSIONS – LEGEND

Dimension	Description	Standard	
		mm	thou
A	Line width	0.1524	6
B	Legend to pad	No clearance required.	
C	Legend to hole	No clearance required.	

Colours available as standard: white. On request: Blue, black, red, green.

SURFACE FINISHES

Lintek can offer a variety of surface finishes to suit any requirement. Electroplated finishes can be tailored to suit various applications. Please contact Lintek to further discuss surface finish options.

Finish	Description	Thickness range (microns)
HASL - Hot air solder levelling (63/37-Sn/Pb)	Levelled with 0° air knives for smoother finish.	25
Electroplated nickel	Using a nickel sulfamate process, 99.9% pure nickel provides an abrasion, corrosion and wear resistant finish.	0.5 – 5
Electroplated gold – soft	99.9% pure gold, providing a highly conductive surface and corrosion resistant finish ideal for all classes of wire bonding.	0.1 – 3
Electroplated gold – hard	Hardened with cobalt, hard gold provides a high conductivity, abrasion resistant contact finish.	0.1 – 2
Electroplated silver	Lower cost than gold, electronics grade electroplated silver provides a bright finish and low contact resistance.	0.1 – 10
Immersion gold	A thin, high purity gold finish that provides a high conductivity and solderable surface.	0.05 – 0.15
Immersion silver	A thin, low cost, highly solderable finish that is the preferred RoHS finish.	0.1 – 0.2
ENIG – Electroless Nickel Immersion Gold	Thin Immersion gold finish over thick Nickel surface barrier providing highly solderable surface.	4-6

QUALITY ASSURANCE

We offer several options for quality assurance.

Electrical testing of finished products guarantees the integrity of inner layers and fine tracks. Minimum pad size for testing is 100 microns, with a minimum pitch of 350 microns. Tests are performed for both continuity and isolation.

We additionally offer manual inspection, Certificate of Conformance documents and First Articles of Inspection Reporting. Where required, we will prepare encapsulated coupon for inspection reports to confirm copper and surface finish plating thickness, etch factor of conductors, and other physical characteristics of the product.

Lintek has IPC-A-600/6012D certified staff and an IPC-A-600 trainer on site. All PCBs are manufactured to IPC Class 2 minimum. Certification is available on request.

Additionally, we are compliant with Quality System AS/NZS ISO 9001:2015 + AS9100D. Subject to material and process constraints, we can produce boards with Underwriters Laboratories (UL) Approval (File number E124884).

MANUFACTURING LEAD TIMES

Regarding delivery, we suggest that our clients consider our standard lead times. As can be expected, small run prototypes have a faster lead time than full scale production.

Board Type	Standard Lead Time (working days)	
	Prototypes	Production
Single sided	7	10
Double sided, with plated through holes	10	12+
Multilayer (layer count dependent)	12-15	15+
Metal backed	15	20+

These lead times are a guide only and may vary depending on complexity and factory conditions. Please contact us for the latest information and an accurate estimate of our delivery capabilities.

In addition to our standard lead times, we offer a rush service for an additional fee – if capacity allows.

Please contact us for details.

LINTEK MINI PROTOTYPE SERVICE

Lintek's Mini Prototype service is specifically designed to meet a requirement for a small, very fast turnaround, single- or double-sided PCB. Instead of requiring a full-sized standard panel for production, we can offer fast processing and delivery on a 280mm x 200mm panel for $350 + GST.

In order to deliver this product at a low price, the board design must comply with the restrictions below.

To allow our CAD engineers time to set up each panel before production begins, purchase orders and files must be received before 9am on a Monday. Production will commence at 9am every Tuesday, and the panels will be dispatched after 3 working days.

Characteristic	Value
PCB file format	RS274X or Protel PCB files, multiple designs should be sent as one file
PCB Type	Double sided with PTH, double sided with NPTH, or single sided
Material	1.6mm FR4
Usable area	280mm x 200mm
Maximum boards per panel	25
Separation between boards	10mm
Copper thickness	30µm – 50µm
Minimum drill diameter	0.4mm
Minimum router diameter	2.0mm
Minimum track and space width	175µm
Minimum annular ring	200µm (after electroplating)
Solder mask	Both sides, green only, minimum of 100µm clearance around pads
Overlay/silkscreen	Both sides, white, minimum feature width of 150µm
Board layout	Drilled tabs
Finish options	HASL(tin/lead) or Immersion Silver
Electrical testing	Optional, additional $50 + GST
Shipping	Within Australia, $25 + GST

JOB INFORMATION

DRAWINGS

Design drawings should be included with any new design. The drawings should clearly show all dimensions and hole sizes, with different hole sizes clearly represented to allow quick inspection.

Holes can be marked as plated, non-plated, blind, buried, filled or unfilled, plugged and capped, tapered, and back drilled. Marked hole diameters are assumed to be finished diameter (after plating) unless otherwise specified.

Drawings should include important mechanical dimensions, including overall board dimensions, slots, plated slots, edge plating, cavities, cut outs, split levels, castellations, and notches.

The following specifications should also be provided, either on the design drawing or in a separate file:

- board material
- board thickness
- copper plating thickness
- minimum trace width
- minimum clearance
- number of layers
- surface finish
- solder mask colour
- legend colour
- IPC Class (IPC-A-600 class 2 is default)
- quality assurance requirements such as coupons, first article of inspection, bare board testing
- for multilayer boards – copper thickness of internal layers, materials and layup

DESIGN FILES

Preferred format for design files is ODB++ or Protel.pcb format. We support most file types including those generated by the latest version of Altium Designer.

Gerber files are also accepted, but a specification of the file format is required, including:

- data format
- imperial/metric
- absolute/incremental position coordinates
- character encoding

Gerber files should be provided with a corresponding drill file in Excellon format with:

- ASCII character encoding
- absolute position coordinates
- 2,4 Imperial
- omit trailing zeros

Gerber files are to be identified using the standard Altium/Protel extensions.

Extension	Layer	Description
GBL	Bottom layer	Copper layer
GBO	Bottom overlay	Overlay, silkscreen, legend
GBP	Bottom paste	Solder paste for SMD assembly (not offered by Lintek)
GBS	Bottom soldermask	Solder mask, solder resist
GKO	Keep out	Areas for copper exclusion
GM1, GM2..	Mechanical 1, 2..	Mechanical layers
G1, G2..	Midlayer 1, 2..	Internal copper signal layers
GP1, GP2..	Plane 1, 2..	Internal planes for power and ground
GPB	Pads Bottom	Unused
GPT	Pads Top	Unused
GTL	Top Layer	Copper layer
GTO	Top Overlay	Overlay, silkscreen, legend
GTP	Top Paste	Solder paste for SMD assembly (not offered by Lintek)
GTS	Top Soldermask	Solder mask, solder resist

Multiple designs are to be sent as a single PCB file.

Include outlines and cutouts on mechanical layer 1.

Multilayer boards should include a text document indicating the layup order.

APPENDIX 1 – PROCESS COMPARISON