PRINTED CIRCUIT BOARDS

PROCESS CAPABILITIES AND DESIGN FOR MANUFACTURE GUIDELINES

VERSION 2.7
RELEASED 04/05/2016

TABLE OF CONTENTS

INTRODUCTION

Lintek Pty Ltd is an Australian company that has specialised in producing microwave and high frequency printed circuit boards for 30 years. Our unique vacuum deposition 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.

This document serves a dual purpose. The first section outlines Lintek's manufacturing and process capabilities, including standard board types and typical manufacturing tolerances. While we specialise in producing boards for microwave and high frequency applications, we have a wide range of production capabilities that suits any industry. In many cases, we are also able to tighten our process tolerances if you have specific requirements.

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.

Contact **sales@lintek.com.au** for any sales enquiries or for further information on Lintek's process capabilities.

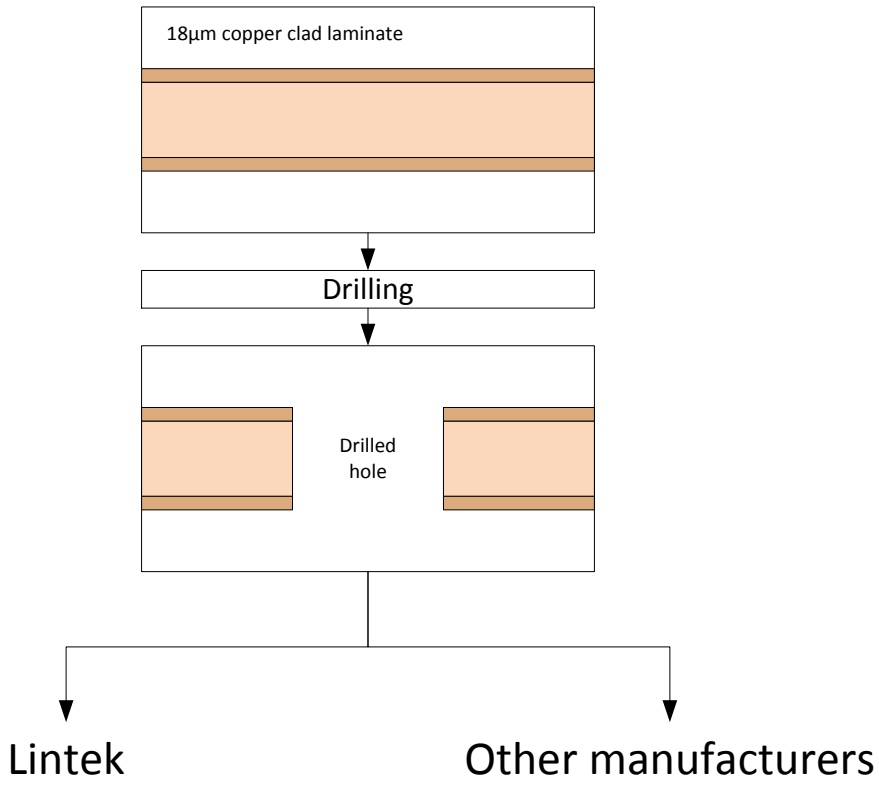
THE LINTEK DIFFERENCE

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 – we deliver copper features that match their designs.

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.

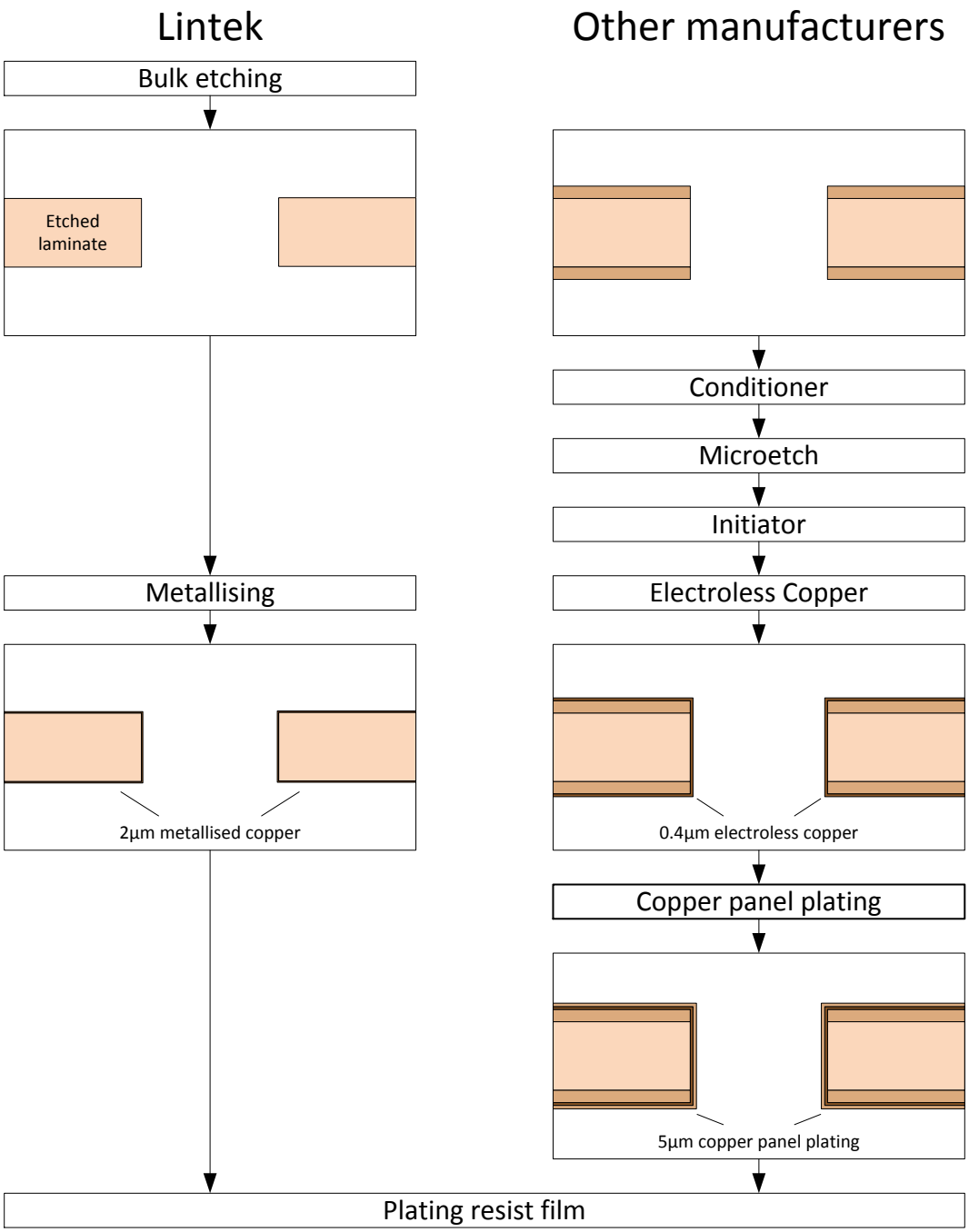
Our method requires fewer process steps, uses less energy and chemicals than traditional manufacturing methods, making Lintek's process significantly more environmentally friendly.

Our manufacturing process starts like other manufacturers. We drill holes in a copper clad laminate, usually starting with 18µm of copper on the panel.

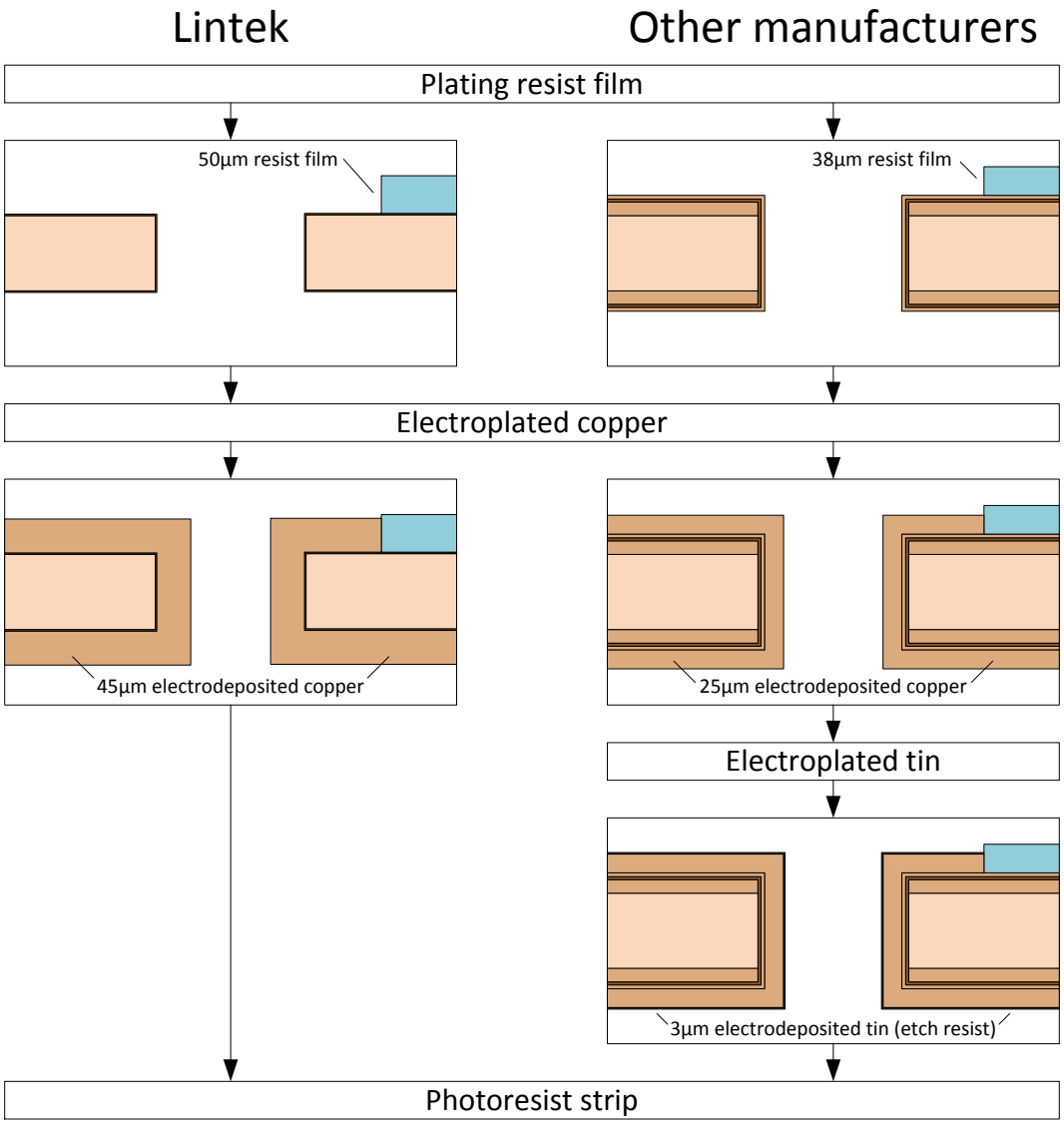


At this point, most manufacturers apply an electroless copper seed layer to enable hole plating. This is the first of several layers of copper that they will add before stripping back a large portion at the end of the process. Electroless copper plating requires surface preparation and often involves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

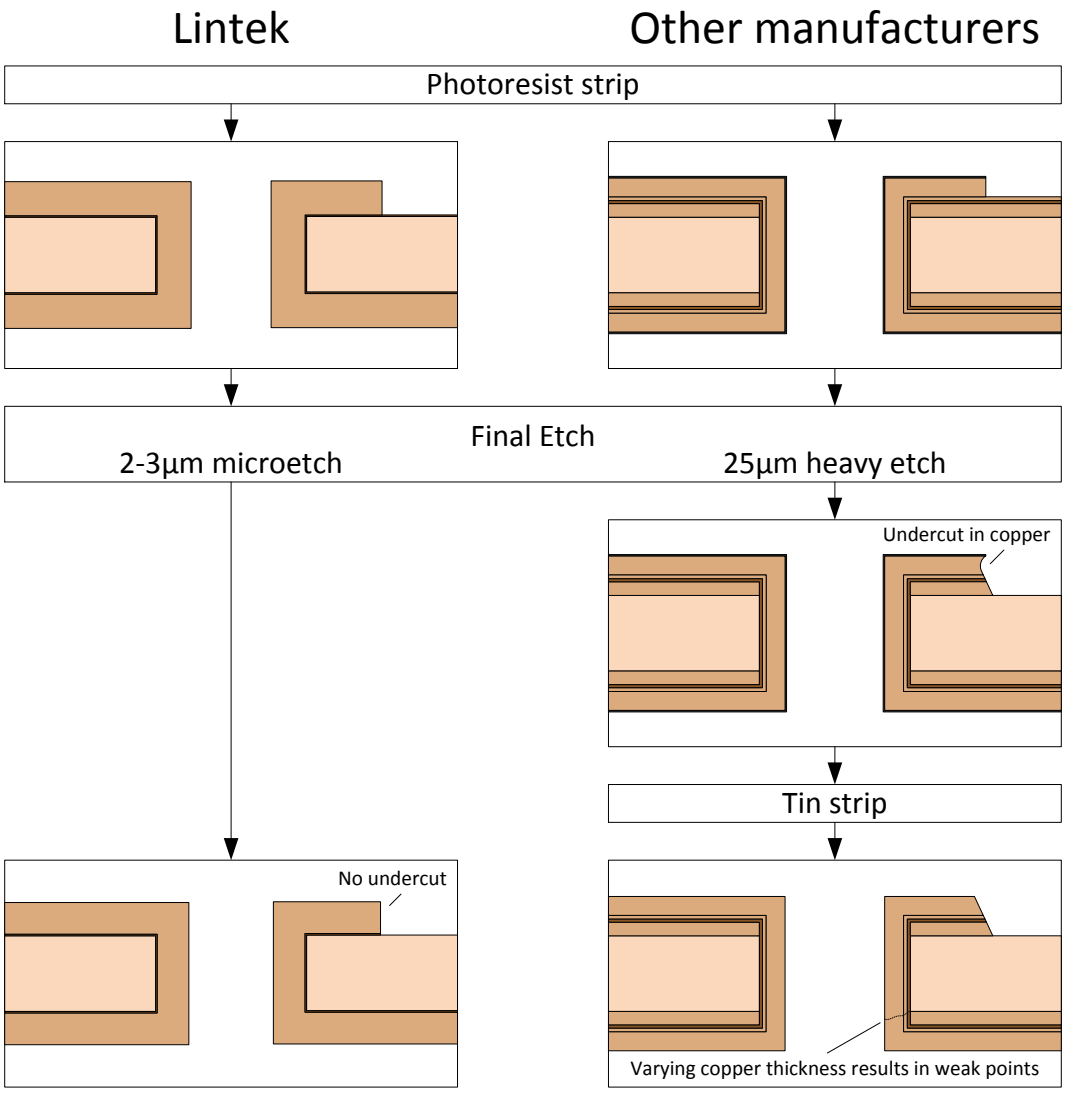


With this seed layer applied, we can then pattern plate the features on the panel with a standard copper electroplating process. After applying a photoimageable plate resist layer, Lintek only requires one process that electroplates 45µm of copper onto our seed layer. Other manufacturers also need to electroplate a tin etch resist layer 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 microetch at this point, removing the 2µm of copper to strip away the seed layer. Stripping such a small amount of copper means that no etch resist is required.

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 sidewall etching. 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.

Our simpler process reduces chemical usage, costs and outputs a superior product. 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
- multilayers with up to 12 layers, including mixed dielectrics
- HDI multilayers with laser drilled micro 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

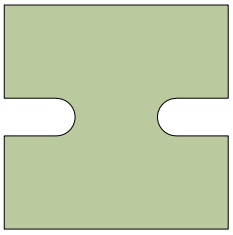
Board Type	Maximum Panel Size		Maximum Usable Area		Maximum Thickness (mm)
	in	mm	in	mm	
Single Sided	18×24	457×610	17×23	432×584	3.2*
Single Sided (Long)#	24×72	610×1829	23×71	584×1803	3.2*
Double Sided	18×24	457×610	17×23	432×584	3.2*
Double Sided (Long)#	24×72	610×1829	23×71	584×1803	3.2*
Double Sided 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 can be produced up to 1800mm in length, without plated through holes, solder mask or legend.

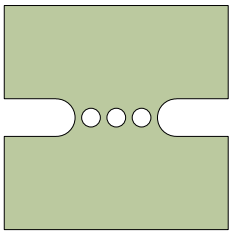
BOARD LAYOUT

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



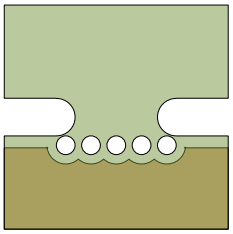
ROUTED TAB

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



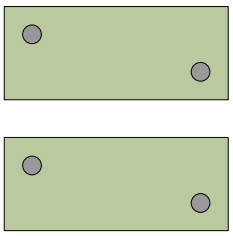
DRILLED TAB (STANDARD)

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.



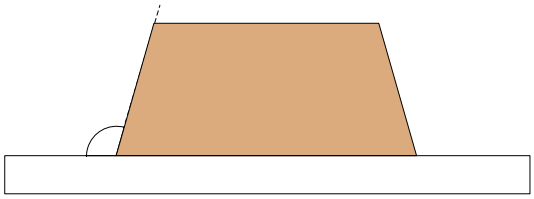
PINNED ROUTE

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

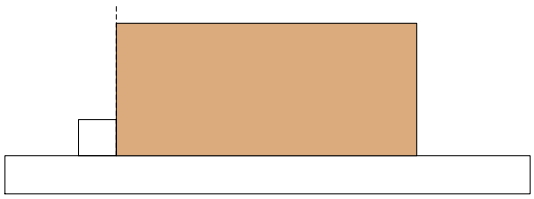
PROCESS TOLERANCES

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.



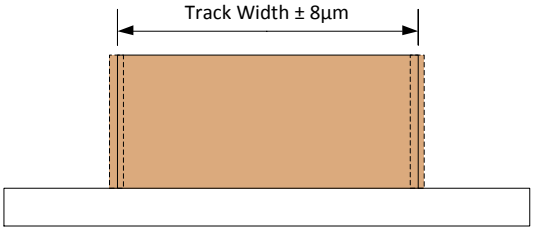
Typical profile of etched conductor

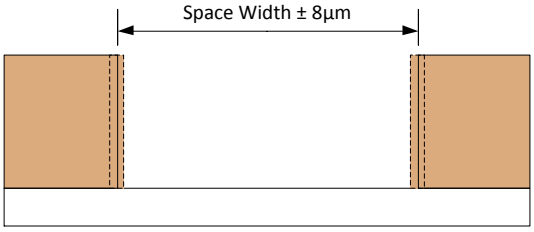


Straight walls produced by Lintek process

The standard tolerance on the width of all electroplated tracks and spaces is ±0.012mm.

The special tolerance on the width of all electroplated tracks and spaces is ±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

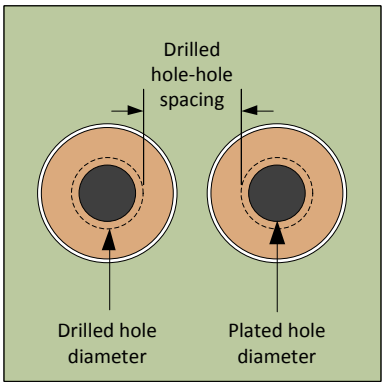
Lintek prefers to have a minimum of one ounce electroplated copper for all Plated Through Hole applications. The copper plating has a finished thickness of 35^{+10}_{-0}microns. Very accurate track and space with minimal sidewall undercut is also achievable with copper traces up to 100 microns thick.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

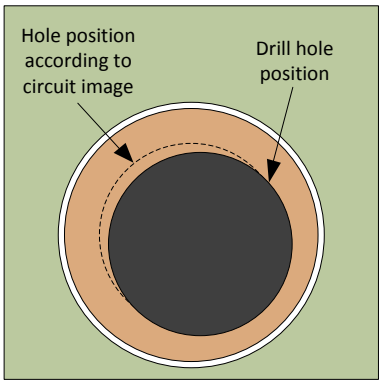
DRILLING

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 (dependent on laminate)	200 microns
Controlled depth drilling precision	±75 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 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

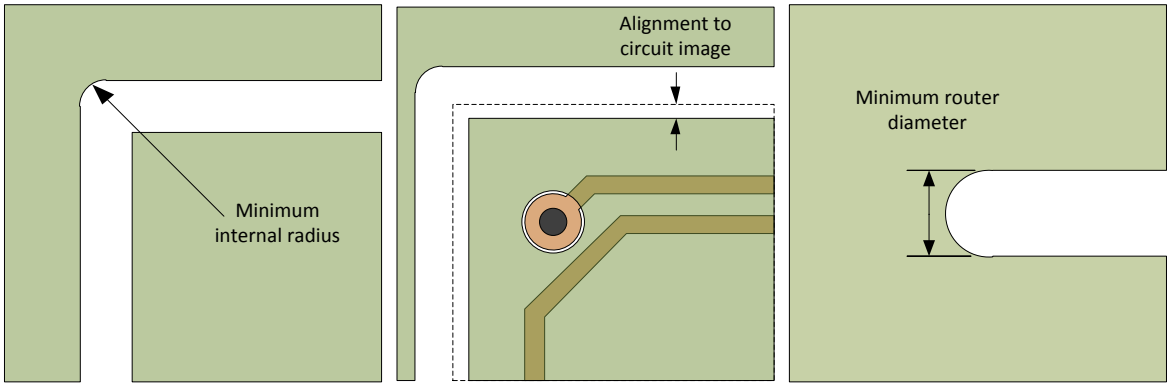
A further ±0.03 is added for substrates thicker than 1.6mm.

ROUTING

Lintek's CNC routers also maintain excellent process tolerances.

Characteristic	Value
Profile feature 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 repeatability	±20 microns
Minimum drilled hole-hole spacing (dependent on laminate)	100 microns
Controlled depth milling	±50 microns
Profile feature tolerance	±50 microns
Minimum internal radius of routed area	0.25mm
Milled surface roughness (R_a)	80 microinches (STD)

LASER DRILLING/MACHINING

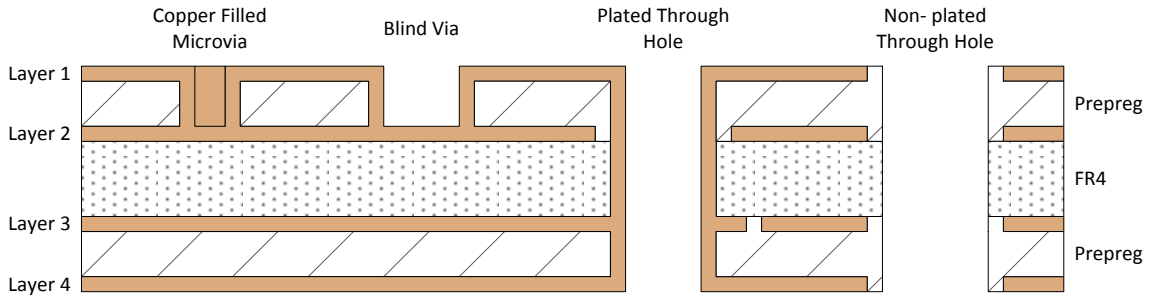
Laser drilling and profiling allows accurate machining not achievable when using mechanical drilling or profiling equipment. Lintek has developed laser machining capabilities for a wide range of exotic PTFE and standard FR4 substrates. Please contact sales@lintek.com.au for information on compatible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

Characteristic	Value
Hole positioning accuracy	±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)	50-200 microns
Controlled depth drilling accuracy to buried copper layer	±5 microns
Deepest hole	500 microns

COPPER PLATING BLIND VIAS

Blind vias to be copper plated require a minimum aspect ratio of 1:1 (Drill diameter = Hole depth). Lintek can also electroplate and copper fill blind microvias with drilled hole diameters of ≤200 microns. It is possible to conformally plate larger diameter blind vias with copper, as long as the aspect ratio is maintained at a minimum of 1:1, or with a hole diameter that is greater than the hole depth.

Please refer to the later section *Pth Aspect Ratio* to ensure that all holes observe our aspect ratio requirements.



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 10 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.

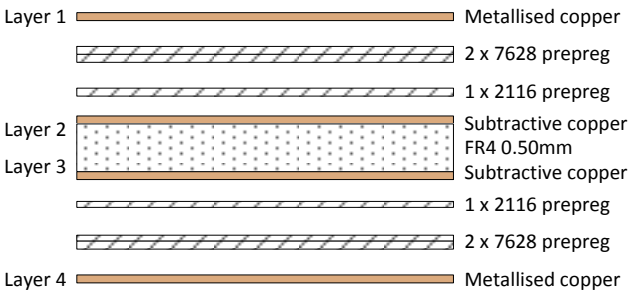
We typically use our proprietary metallising technology to produce excellent impedance characteristics on the outer layers of a board, and use low-cost subtractive layers for internal power and ground planes. We are able to include metallised inner layers for controlled impedance features if required.

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. Choosing a standard layup for your board design will accelerate the release to manufacturing, as these layup configurations have been regularly used and thoroughly tested.

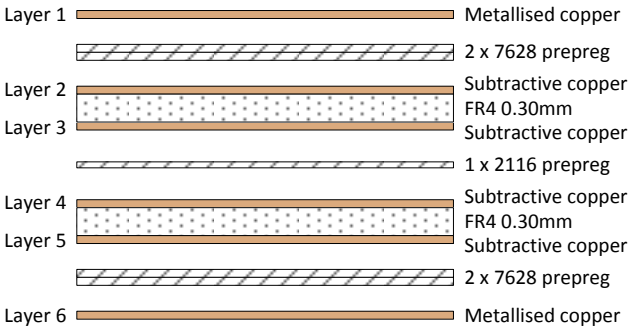
If you are considering a more complex design or a multilayer that includes mixed dielectrics, you are welcome to contact us during the design phase of your board and we will assist in ensuring the manufacturability of your design.

4 LAYER BOARD



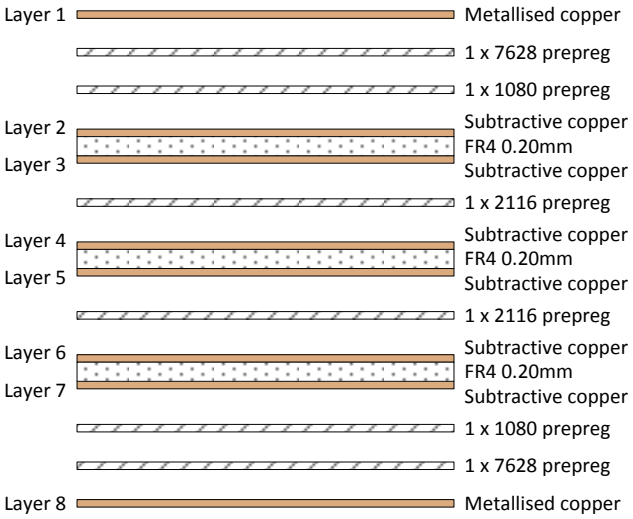
Finished Thickness: 1.556mm

6 LAYER BOARD



Finished Thickness: 1.612mm

8 LAYER BOARD



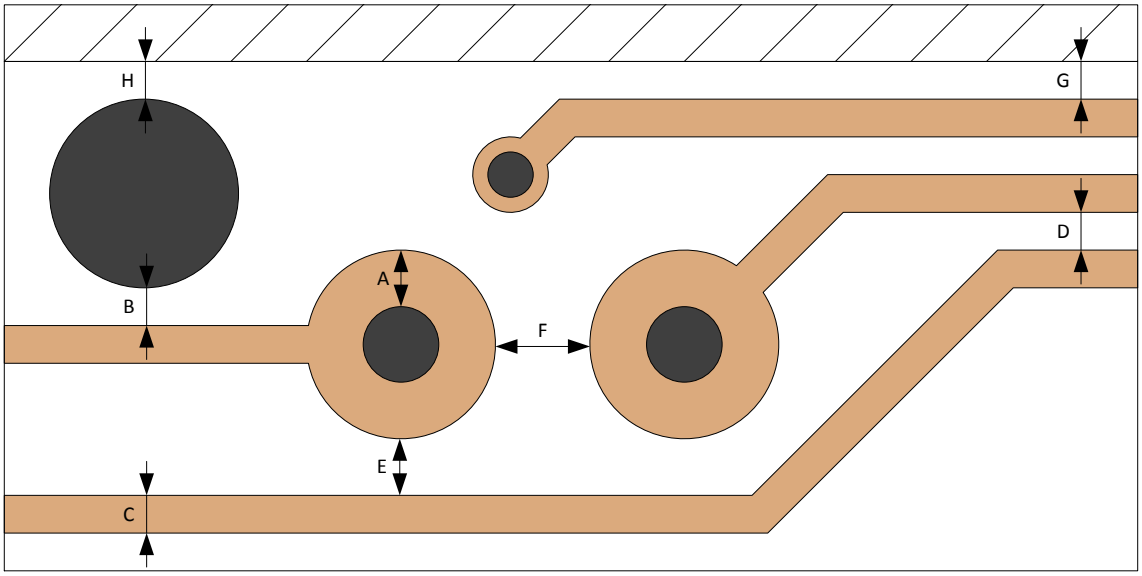
Finished Thickness: 1.584mm

The properties of our commonly used prepregs are shown below to aid impedance calculations.

Material	Cured Thickness (mm)	Testing Frequency (Ghz)	D_k	D_f
1080	0.066	0.1	4.5-4.8	0.020-0.030
2116	0.114	0.1	4.5-4.8	0.020-0.030
7628	0.172	0.1	4.5-4.8	0.020-0.030
High Tg 1080	0.075	0.1	4.3	0.016
High Tg 2116	0.121	0.1	4.3	0.016
High Tg 7628	0.177	0.1	4.3	0.016
Low flow 1080	0.069	0.1	4.3	0.025
Rogers 4450B	0.10	10	3.54	0.004
Mercurywave 9350	0.10	10	3.5	0.004
*AR6700 Bondfilm	0.04	10	2.35	0.0025

*note: AR6700 is a thermoplastic bonding film which has Crystalline Melt Point of 184 degrees C.

EXTERNAL LAYERS



MINIMUM DIMENSIONS – SIGNAL LAYERS

Dimension	Description	Standard		Special	
		mm	thou	mm	thou
A	Hole to pad (annular ring)*	0.203	8	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 annular ring is based on finished hole diameter after electroplating PTH barrel.

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

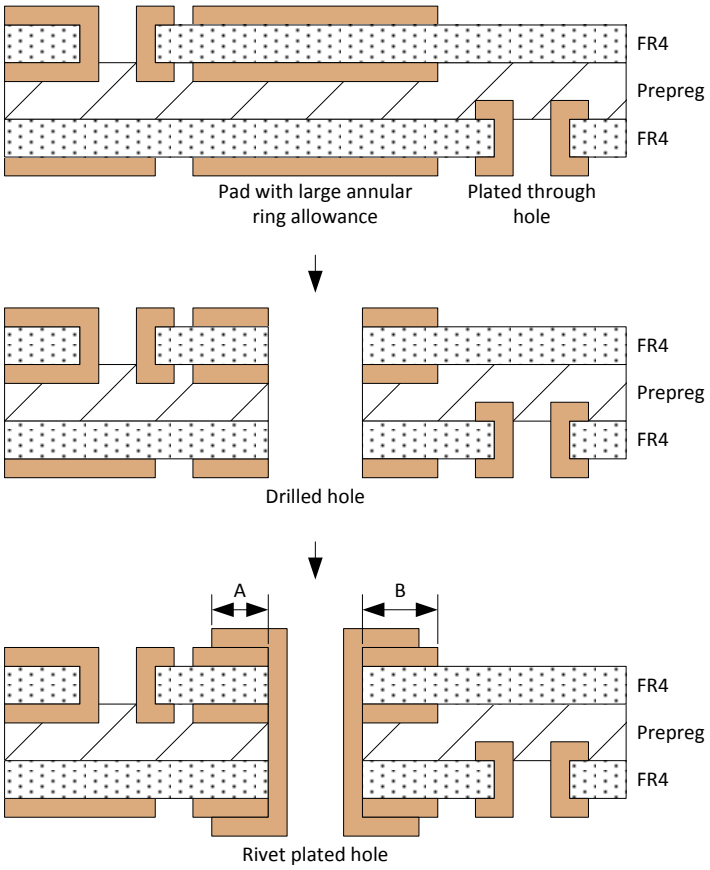
RIVET PLATING (COPPER WRAP)

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.

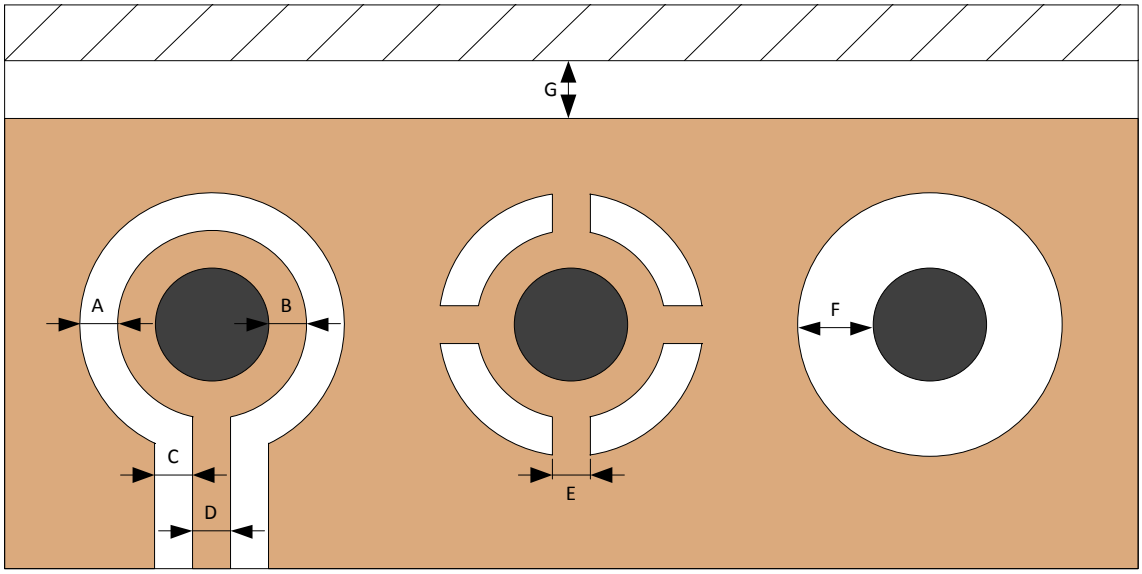
The diagram below shows the process of rivet plating and a typical cross section of the finished product.



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

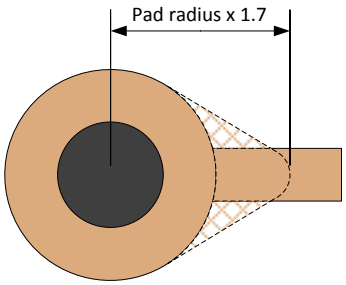
INTERNAL POWER AND GROUND PLANES



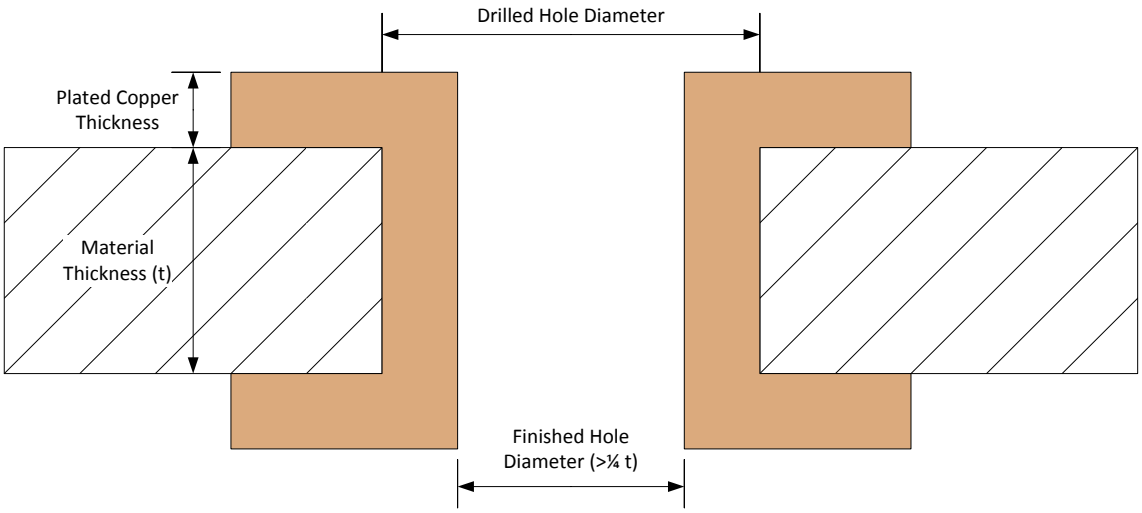
MINIMUM DIMENSIONS – POWER AND GROUND PLANES

Dimension	Description	Standard		Special	
		mm	thou	mm	thou
A	Plane to pad	0.2032	8	0.1016	4
B	Hole to pad	0.3048	12	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	0.3810	15	0.1016	4
F	Plane to hole	0.3048	12	0.1905	7.5
G	Plane to board edge	0.5080	20	0.1524	6

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

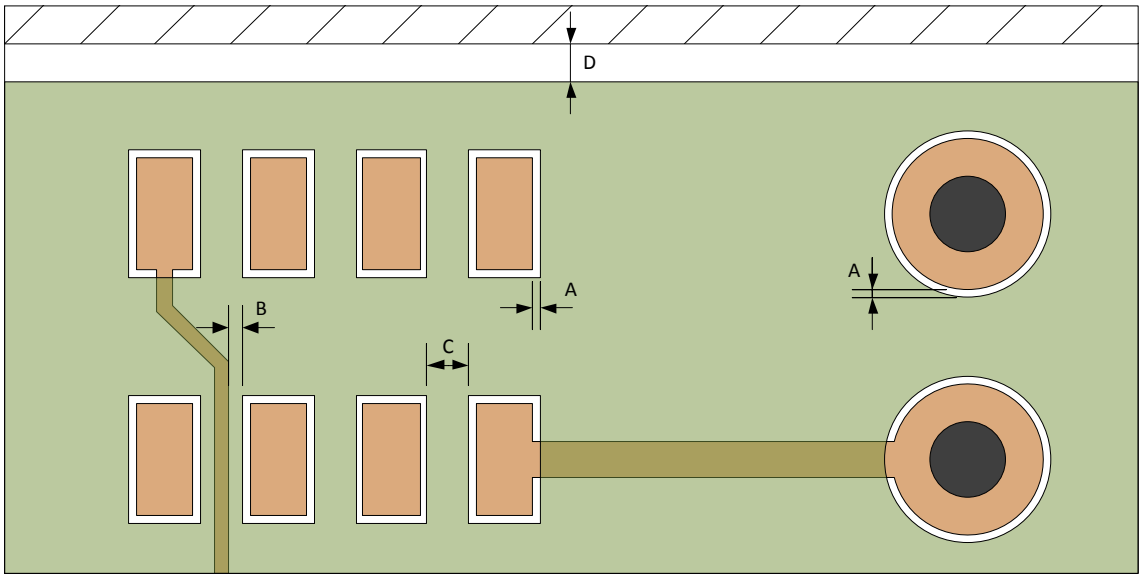


Lintek is able to produce plated vias with a standard aspect ratio of 4:1. As such, the drilling diameter of any plated through hole must be greater than ¼ the thickness of the layer/s it passes through.

For instance, on a 1.60mm thick substrate, the minimum drilling diameter is 0.4mm. This will provide a 0.33mm finished hole, assuming one ounce (35um) of copper plating.

For applications where an aspect ratio of greater than 4:1 is required, please contact sales@lintek.com.au to discuss possible options.

SOLDER MASK



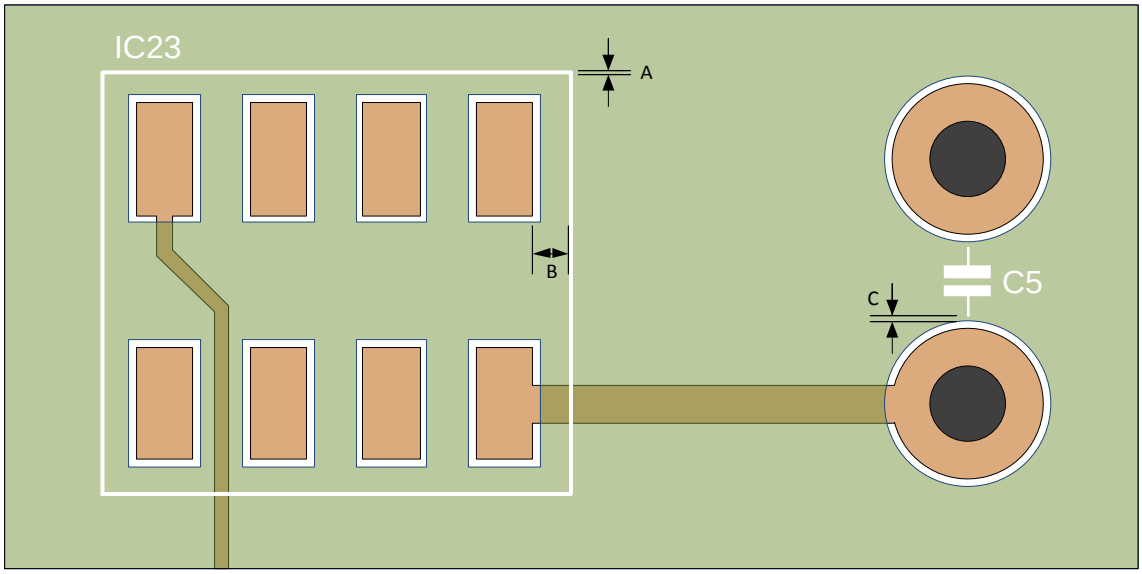
MINIMUM DIMENSIONS – SOLDER MASK

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 is able to offer a variety of surface finishes to suit any requirement.

Finish	Description	Thickness range (microns)
Hot air solder levelling	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 provides a highly solderable, conductive finish. Compatibility is material dependent.	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 for wire bonding.	0.05 – 0.15
Immersion silver	A thin, low cost, highly solderable finish that is the preferred RoHS finish.	0.1 – 0.2

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 A600 certified staff and a 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 9001:2008. Subject to material and process constraints, we are able to 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. Typically small run prototypes have a faster lead time than full scale production.

Board Type	Standard Lead Time (working days)	
	Prototypes	Production
Single sided	5	7
Double sided, with plated through holes	7	10
Multilayer (layer count dependent)	10	12-15
Metal backed	12	15

These lead times are a guide only, and may vary depending on 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. 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 $290 + GST.

In order to deliver this product at such 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 , $15 + GST

JOB INFORMATION

DRAWINGS

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

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, cut outs 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
- surface finish
- solder mask colour
- legend colour
- 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 Protel.pcb format. We support files generated by any software up to Altium Designer 13.

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

Extension	Layer	Description
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