

MINIMUM ARTWORK FEATURE SIZE

Track to track:	n/a	mm	n/a	inches
Track to pad:	n/a	mm	n/a	inches
Track width:	n/a	mm	n/a	inches
Drill Hole Size:	2mm	mm	0.079	inches

BOARD TYPE

- ☒ Single Layer
- ☐ Double layer
- ☐ Multi Layer,
- ☐ HDI Multi Layer
- ☐ Embedded PCB Resistor

VIA CONSTRUCTION

- ☐ Blind Vias
- ☐ Buried Vias
- ☐ HDI Micro Vias
- ☐ Stacked HDI Micro Vias

GENERAL NOTES:

1. Finished Board Thickness and Layer Stackup shown in View 'A'
2. All gerber files are in RS274X Metric 4:4 format with embedded apertures.
3. Board shall be fabricated as per IPC-6012B guidelines (unless otherwise stated).
4. ~~Laser vias are to be Copper filled~~
5. ~~Mechanical vias are to be epoxy filled~~
6. All hole sizes described in this Document are Finished Hole Sizes (FHS).
7. ~~No silkscreen allowed on exposed lands or in holes.~~
8. Manufacturer to add their ID/Logo and Datecode on Top Overlay or Top Solder Mask, as Appropriate.
9. Manufacturer to add UL Flammability Rating on Top Overlay or Top Solder Mask, as Appropriate.
10. Finished boards shall not have nicks, scratches, voids, poor plating or mis-drilled holes
11. The PCB is to be 100% Electrically Tested. No Cross-Outs allowed on panel
12. Lamination stack-up must be submitted for approval prior to PCB manufacture
13. No copper to be added or removed from the design, except that which is stated above.

MATERIALS

PCB / Panel Size: 48.0x12.5mm

PCBs Per Panel:

Laminate : FR4

Thickness: 0.8mm +/- 10% After Plating

Flammability : UL94-V0

PROFILE METHOD

- ☒ Routing, 2mm Rout Path
☐ V-Score 30 Deg

ACCEPTABILITY

- ☐ IPC-3-300
- ☒ IPC-A-600 Class 2
- ☐ IPC-A-600 Class 3
- ☒ UL Approval

SURFACE MOUNT

- ☐ Single Sided
- ☐ Double Sided

SOLDER MASK :

SMOBC LPI GREEN

- Top Solder Mask
 □ Bottom Solder Mask

SCREEN PRINT OVERLAYS:

- ☐ Top Overlay (White)
- ☐ Bottom Overlay

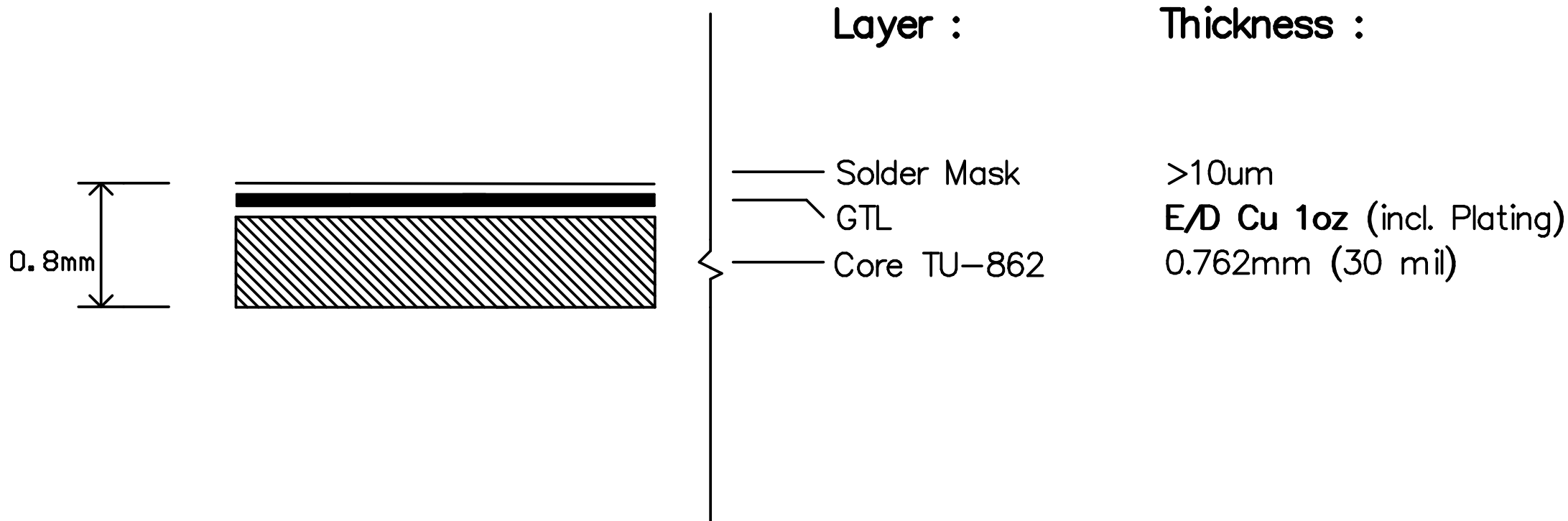
TRACK FINISH:

- ☐ Hot air levelled
- ☒ Immersion Gold over Nickel (1u")
- ☐ OSP (Organic Solder Preservatives)
- ☐ Silver Immersion (6-20u")
- ☐ Hard Gold

PRODUCTION FILE LIST

	DESCRIPTION	FILE
↓ ⇒ ⇒ LAMINATION SEQUENCE ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒ ⇒	<input type="checkbox"/>	
	<input type="checkbox"/> TOP OVERLAY	GTO
	<input checked="" type="checkbox"/> TOP SOLDERMASK	GTS
	<input checked="" type="checkbox"/> TOP COPPER LAYER	GTL
	<input type="checkbox"/> BOTTOM COPPER LAYER	GBL
	<input type="checkbox"/> BOTTOM SOLDERMASK	GBS
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input type="checkbox"/>	
	<input checked="" type="checkbox"/> BOARD OUTLINE	GM1
	<input checked="" type="checkbox"/> ROUTE OUTLINE	GM2
	<input type="checkbox"/> V—SCORE	GM3
	<input type="checkbox"/> PASTE SHIM OUTLINE	GM14
	<input type="checkbox"/> TOP PASTE MASK	GTP
	<input type="checkbox"/> BOTTOM PASTE MASK	GBP
	<input checked="" type="checkbox"/> APERTURE FILE	APR
	<input checked="" type="checkbox"/> DRILL DRAWING	GDx
	<input checked="" type="checkbox"/> DRILL GUIDE	GGx
	<input checked="" type="checkbox"/> NC DRILL — TOOL FILE	DRL
	<input checked="" type="checkbox"/> NC DRILL — TEXT FILE	TXx
	<input checked="" type="checkbox"/> NC DRILL — BINARY FILE	DRx

VIEW 'A' : LAMINATION STACK-UP STRUCTURE



NOTES:

1. Finished PCB thickness tolerance $\pm 10\%$
2. Finished PCB board outline tolerance $\pm 0.1\text{mm}$

CONTROLLED IMPEDENCE

None

DRILL TABLE

Symbol	Count	Hole Size	Plated	Hole Type
O	2	78.74mil (2.000mm)	NPTH	Round
	2 Total			

Windcave Ltd