Lime microsystems

LimeNet-Micro v2.1

CONTROLLED IMPEDANCE

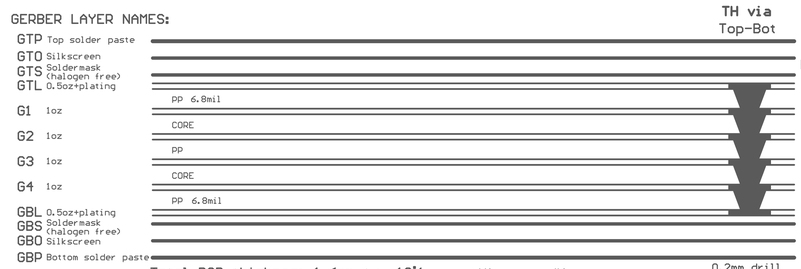
GENERAL PARAMETERS

Parameter	Layer/Layers	Value
Copper foil thickness	Top	17.5um
Dielectric thickness between layers	Top-L2	173um (6.8 mils)
Dielectric permittivity between layers (Er)		4.2
Copper foil thickness	Bottom	17.5um
Dielectric thickness between layers	Bottom-L11	173um (6.8 mils)
Dielectric permittivity between layers (Er)		4.2

CALCULATIONS

RF (Top)	Target impedance	Single-ended	50R
	Additional comments	Top layer, no side GND plane	
	Top layer copper foil thickness	17.5 um	
	Track width	0.325 mm (12.795 mils)	
	Top layer copper foil thickness	173um (6.8 mils)	
	Dielectric thickness between layers	Top-L2	173um (6.8 mils)
	Dielectric permittivity between layers (Er)		4.2
	Approximate microstrip line impedance	49.99 Ohms (+/- 10% tolerance)	
RF (Top)	Target impedance	Differential	100R
	Additional comments	Top layer, no side GND plane	
	Top layer copper foil thickness	17.5 um	
	Track width	0.2 mm (7.874 mils)	
	Track spacing	0.14 mm (5.511 mils)	
	Top layer copper foil thickness	173um (6.8 mils)	
	Dielectric thickness between layers	Top-L2	173um (6.8 mils)
	Dielectric permittivity between layers (Er)		4.2
	Approximate microstrip line impedance	100.6 Ohms (+/- 10% tolerance)	
RF (Bottom)	Target impedance	Single-ended	50R
	Additional comments	Bottom layer, no side GND plane	
	Top layer copper foil thickness	17.5 um	
	Track width	0.325 mm (12.795 mils)	
	Top layer copper foil thickness	173um (6.8 mils)	
	Dielectric thickness between layers	Bottom-L5	173um (6.8 mils)
	Dielectric permittivity between layers (Er)		4.2
	Approximate microstrip line impedance	49.99 Ohms (+/- 10% tolerance)	
HDMI (Bottom)	Target impedance	Differential	100R
	Additional comments	Bottom layer, no side GND plane	
	Top layer copper foil thickness	17.5 um	
	Track width	0.2 mm (7.874 mils)	
	Track spacing	0.14 mm (5.511 mils)	
	Top layer copper foil thickness	173um (6.8 mils)	
	Dielectric thickness between layers	Bottom-L5	173um (6.8 mils)
	Dielectric permittivity between layers (Er)		4.2
	Approximate microstrip line impedance	100.6 Ohms (+/- 10% tolerance)	

STACKUP AND LAYER DESCRIPTION



Total PCB thickness: 1.6mm +/- 10%

Via type #1	0.2mm drill 0.4mm ring
Via type #2	0.4mm drill 0.8mm ring
Via type #3 (In pad, IC1, resin filled with metal cap)	0.2mm drill 0.35mm ring
Via type #4 (In IC1 thermal pad, NO resin fill and metal cap)	0.2mm drill 0.5mm ring

VERY IMPORTANT NOTES

GENERAL PCB SPECS

Minimum copper to copper spacing	0.1mm (3.9mil)	
Minimum track width	0.1mm (3.9mil)	
PCB thickness	1.6mm +/-10%	
PCB material	IT-180A preferred or equivalent RF-rated material (up to 3 GHz or higher)	
Copper weight	External layer	0.5 oz+plating
	Internal layer	1 oz
Solder mask	DARK BLUE	
	Both sides	
	Halogen free	
	Glossy finish (NOT matte)	
Silkscreen	White epoxy ink	
	Both sides	
	Halogen free	
	No silkscreen on pads	
Hole types on the PCB and information	Plated ☒	Non-plated ☒
	● Hole diameters are final manufactured diameters INCLUDING HOLE METALIZATION.	
Route process	V-score ☐	Tab route ☒
	V-score and tab route ☐	
Panel	Yes ☐	No ☒
Surface finish (both sides)	HASL lead free ☐	
	HASL with lead ☐	
	Immersion gold ☒	0.05-0.10um of gold over 2.50-5.00um of nickel
	OSP ☐	
	Hard gold ☐	

ADDITIONAL REQUIREMENTS

- All 0.2mm vias including 0.35mm ring and 0.2mm drill via-in-pads (IC1) must be resin filled with metal cap.
- IC1 thermal pad vias with 0.4mm ring and 0.2mm drill must be resin filled with metal cap. IC1 thermal pad vias with 0.5mm ring and 0.2mm drill must be left open (NO resin fill with metal cap). 4 vias in total, marked with note.
- Electrical test : 100 % netlist.
- Boards are to be individually bagged.
- PCB vendor to silkscreen UL and RoHS compliance marks, vendor logo and date code on bottom where shown (ignore if none of the info will be placed on PCB)
- Assembly note: Assembly house MUST provide notes in paper with shipped board if there were any changes during assembly and the board is not assembled 100% according to BOM and P&P files.
  Note example:

Part	Initial BOM asm. note	Current PCB status	Comment
R1	FIT	NF	Not mounted due to bad footprint
IC5	FIT	NF	Not mounted due to part shortage