Lime microsystems

LimeSDR-USB v1.4s

CONTROLLED IMPEDANCE

GENERAL PARAMETERS:

Top layer copper foil thickness: 17.5 um
Dielectric thickness from Top to L2 = 173um (6.8 mils)
Dielectric between Top layer and 2nd layer relative permittivity (Er): 4.2

Bottom layer copper foil thickness: 17.5 um
Dielectric thickness from L11 to Bottom = 173um (6.8 mils)
Dielectric between L11 layer and Bottom layer relative permittivity (Er): 4.2
Ground plane distance to trace on Bottom layer: 0.1mm

CALCULATIONS:

RF (Top)

50 Ohm microstrip (Top layer, no GND) characteristics:
Top layer copper foil thickness: 17.5 um
Track width = 0.325 mm (12.795 mils)
Dielectric thickness from Top to L2 = 173um (6.8 mils)
Dielectric between Top layer and 2nd layer relative permittivity (Er): 4.2

Approximate microstrip line impedance = 49.99 Ohms (+/- 10% tolerance)

RF (Top)

100 Ohm coupled microstrip line (Top layer) characteristics:
Top layer copper foil thickness: 17.5 um
Track width = 0.2 mm (6.8 mils)
Track spacing = 0.14 mm (5.51 mils)
Track width/spacing ratio = 1.428
Dielectric thickness from top to L2 = 173um (6.8 mils)
Dielectric between Top layer and 2nd layer relative permittivity (Er): 4.2

Approximate coupled microstrip line impedance = 100.752 Ohms (+/- 10% tolerance)

RF (Bottom)

50 Ohm coplanar waveguide with GND (Bottom layer) characteristics:
Bottom layer copper foil thickness: 17.5 um
Track width = 0.254 mm (10 mils)
Distance to GND: 0.1 mm (3.937 mils)
Dielectric thickness from Bottom to L11 = 173um (6.8 mils)
Dielectric between Bottom layer and L11 relative permittivity (Er): 4.2

Approximate microstrip line impedance = 49.99 Ohms (+/- 10% tolerance)

USB3.0 (Top)

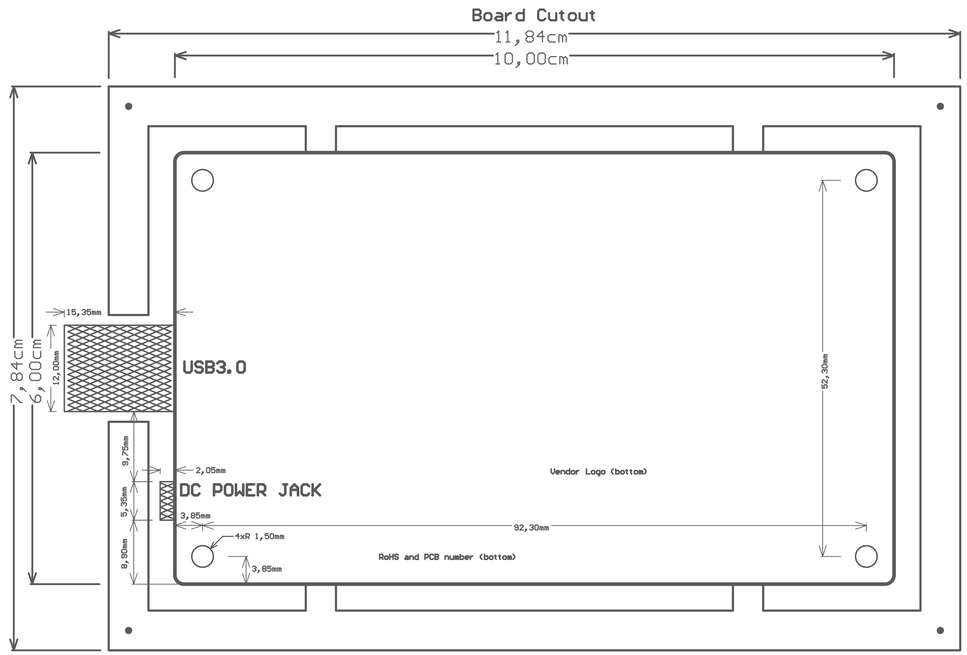
90 Ohm coupled microstrip line (Top layer, without GND) characteristics:
Top layer copper foil thickness: 17.5 um
Track width = 0.2 mm (6.8 mils)
Track spacing = 0.1 mm (3.93 mils)
Track width/spacing ratio = 2
Dielectric thickness from Top to 2nd layer = 173um (6.8 mils)
Dielectric between Top layer and 2nd layer relative permittivity (Er): 4.2

Approximate coupled microstrip line impedance = 90.5 Ohms (+/- 10% tolerance)

USB3.0 (Bottom)

90 Ohm coupled microstrip line (Bottom layer) characteristics:
Bottom layer copper foil thickness: 17.5 um
Track width = 0.2 mm (6.8 mils)
Track spacing = 0.1 mm (3.93 mils)
Track width/spacing ratio = 2
Dielectric thickness from L11 to Bottom layer = 173um (6.8 mils)
Dielectric between L11 layer and Bottom layer relative permittivity (Er): 4.2

Approximate coupled microstrip line impedance = 90.5 Ohms (+/- 10% tolerance)



STACKUP:

GERBER LAYER NAMES:

GTP	Top solder paste		
GTO	Silkscreen		
GTS	Soldermask (halogen free)		
GTL	0.5oz+plating		ELECTRICAL LAYERS: Top: RF/GND
		PP 6.8mil	
G1	0.1oz		L2: GND
		CORE	
G2	0.1oz		L3: PWR/Signal/GND
		PP	
G3	0.1oz		L4: Signal/PWR/GND
		CORE	
G4	0.1oz		L5: PWR/GND
		PP	
G5	0.1oz		L6: Signal/GND
		CORE	
G6	0.1oz		L7: GND
		PP	
G7	0.1oz		L8: Signal
		CORE	
G8	0.1oz		L9: Signal
		PP	
G9	0.1oz		L10: CLK/Signal
		CORE	
G10	0.1oz		L11: GND
		PP 6.8mil	
GBL	0.5oz+plating		Bottom: Signal/PWR/GND
GBS	Soldermask (halogen free)		ADDITIONAL LAYERS:
GBO	Silkscreen		Mechanical 1: Board cutout
GBP	Bottom solder paste		ASM TOP: Assembly top
			ASM BOT: Assembly bottom
			Mechanical 13: Component 3D body

TH via
Top-Bot

Total PCB thickness: 1.6mm +/- 10%

Via type #1 — 0.2mm drill, 0.4mm ring

Via type #2 (In pad, resin filled with metal cap) — 0.2mm drill, 0.35mm ring

VERY IMPORTANT NOTES:

1) 0.35mm ring and 0.2mm drill via-in-pads (IC1) must be resin filled with metal cap
2) Solder mask : DARK BLUE, both sides, halogen free, glossy finish (NOT matte)
3) Silkscreen : white epoxy ink, halogen free, both sides. No silkscreen on pads.
4) DRCs must be run on Gerber files before building boards
5) Hole diameters are final manufactured diameters INCLUDING HOLE METALIZATION.
6) Minimum track spacing: 0.1 mm
   Minimum track width: 0.1 mm
7) There are plated and non-plated holes on the PCB
8) Material:
   IT-180A
   PCB vendor to silkscreen UL and RoHS compliance marks, vendor logo and date code on bottom where shown
   Copper weight: External layers 0.5 oz+plating
   Internal layers 1 oz
9) Electrical test : 100 % netlist.
10) Boards are to be individually bagged.