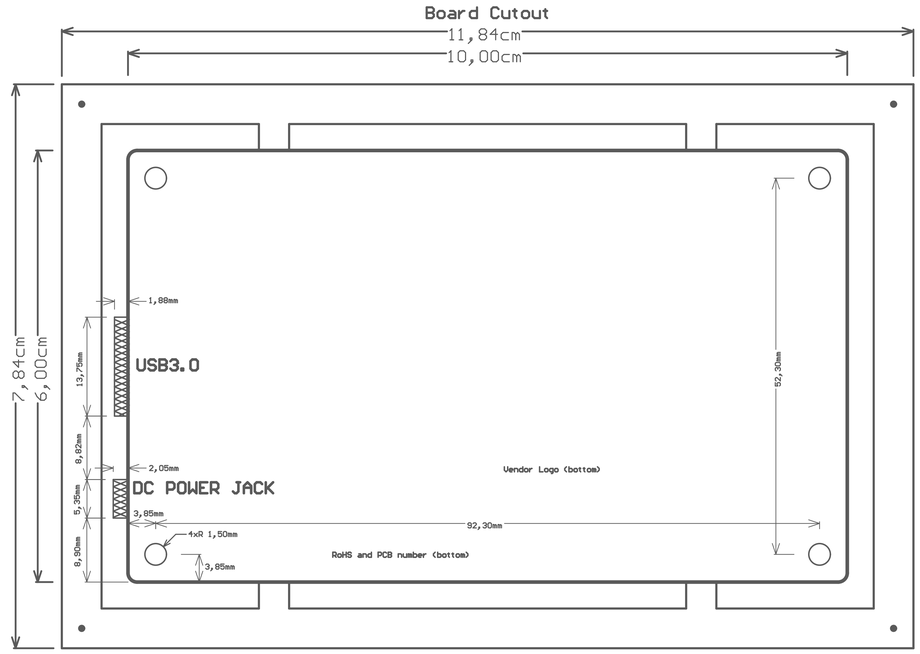
Lime microsystems

LimeSDR-USB v1.4

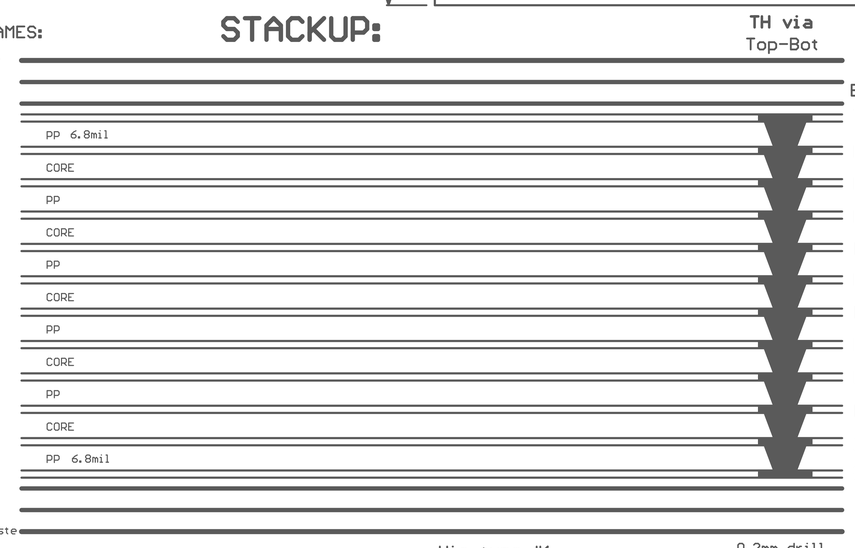
CONTROLLED IMPEDANCE



GERBER LAYER NAMES:

GTP Top solder paste
GTO Silkscreen
GTS Soldermask (halogen free)
GTL 0.5oz+plating
G1 0.1oz
G2 0.1oz
G3 0.1oz
G4 0.1oz
G5 0.1oz
G6 0.1oz
G7 0.1oz
G8 0.1oz
G9 0.1oz
G10 0.1oz
GBL 0.5oz+plating
GBS Soldermask (halogen free)
GBO Silkscreen
GBP Bottom solder paste

STACKUP:



ELECTRICAL LAYERS:
Top: RF/GND
L2: GND
L3: PWR/Signal/GND
L4: Signal/PWR/GND
L5: PWR/GND
L6: Signal/GND
L7: GND
L8: Signal
L9: Signal
L10: CLK/Signal
L11: GND
Bottom: Signal/PWR/GND

ADDITIONAL LAYERS:
Mechanical 1: Board cutout
ASM TOP: Assembly top
ASM BOT: Assembly bottom
Mechanical 13: Component 3D body

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, 0.31mm ring

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)

VERY IMPORTANT NOTES:

1) 0.31mm 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.