# LimePSB RPCM v1.2

## 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-L5 | 173um (6.8 mils) |
| Dielectric permittivity between layers (Er) | | 4.2 |

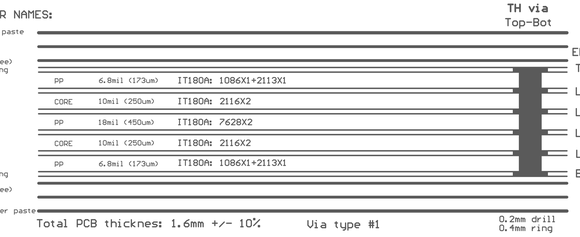
### CALCULATIONS

| | | | |
|---|---|---|---|
| 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) | |
| HDMI (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) | |
| USB3 (Top, Bottom) | Target impedance | Differential | 90R |
| | Additional comments | Top, bottom layer | |
| | Top layer copper foil thickness | 17.5 um | |
| | Track width | 0.2 mm (7.874 mils) | |
| | Track spacing | 0.1 mm (3.94 mils) | |
| | Top layer copper foil thickness | 173um (6.8 mils) | |
| | Dielectric thickness between layers | Top-L2; Bottom-L5 | 173um (6.8 mils) |
| | Dielectric permittivity between layers (Er) | | 4.2 |
| | Approximate microstrip line impedance | 86 Ohms (+/- 10% tolerance) | |

## STACKUP AND LAYER DESCRIPTION

GERBER LAYER NAMES:

GTP Top solder paste
GTO Silkscreen
GTS Soldermask (halogen free)
GTL 0.5oz+plating
G1 1oz (35um)
G2 1oz (35um)
G3 1oz (35um)
G4 1oz (35um)
GBL 0.5oz+plating
GBS Soldermask (halogen free)
GBO Silkscreen
GBP Bottom solder paste




Total PCB thickness: 1.6mm +/- 10%  Via type #1

ELECTRICAL LAYERS:
Top: RF//PWR/GND
L2: GND
L3: Signal/PWR/GND
L4: PWR/Signal/GND
L5: GND
Bottom: RF/Signal/PWR/GND

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

## VERY IMPORTANT NOTES

### GENERAL PCB SPECS

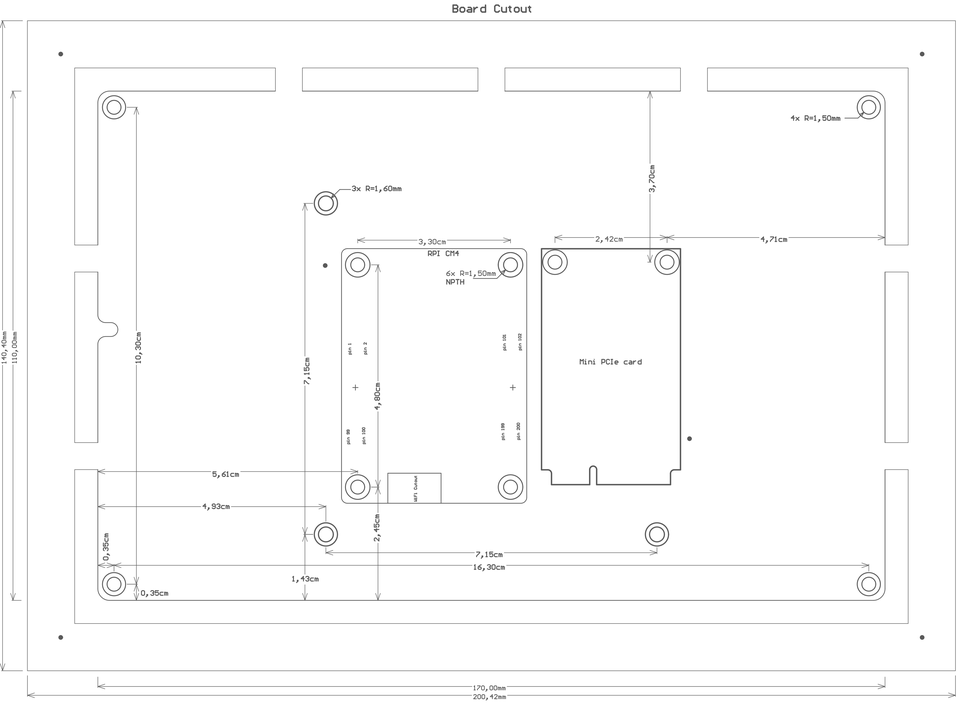
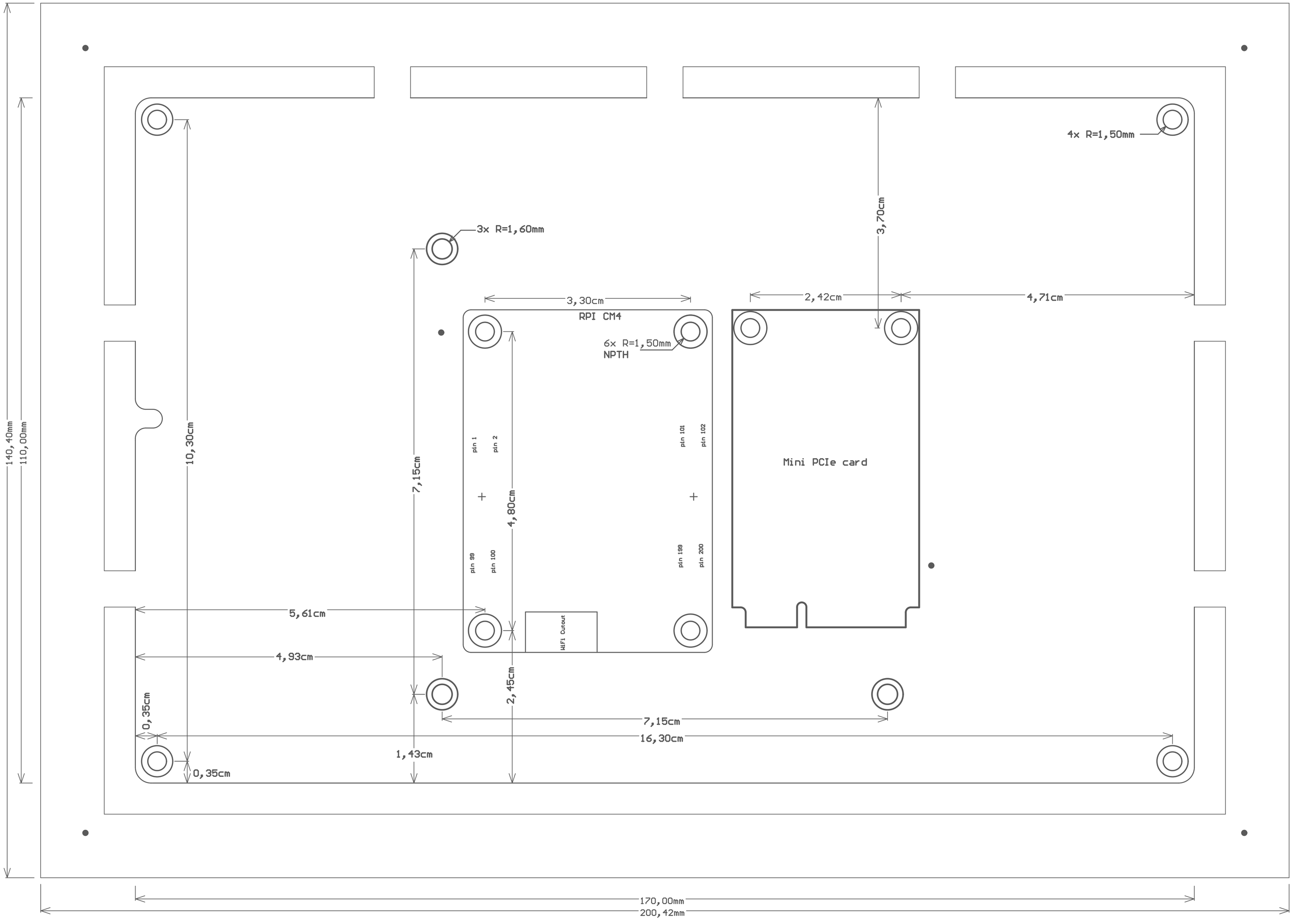
| | | |
|---|---|---|
| Minimum copper to copper spacing | 0.1mm (3.9mil) | |
| Minimum track width | 0.1mm (3.9mil) | |
| PCB thickness | 1.6mm +/-10% | |
| PCB material | IT-180A preferred or equivalent RF-rated material (up to 3 GHz or higher) | |
| Copper weight | External layer | 0.5 oz+plating |
| | 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

- 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 |

Board Cutout



# Board Cutout