ATREB215-XPRO-A PCB Specification

Table of Contents

1 General information

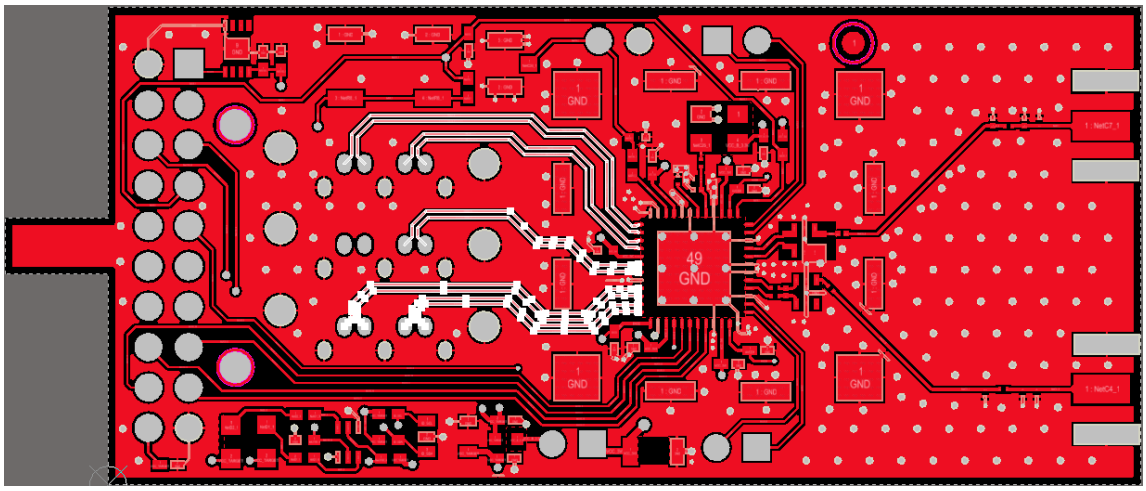
1.1 Board identification

Name: ATREB215-XPRO-A Extension Board
Board identification number: A08-2463 Rev1

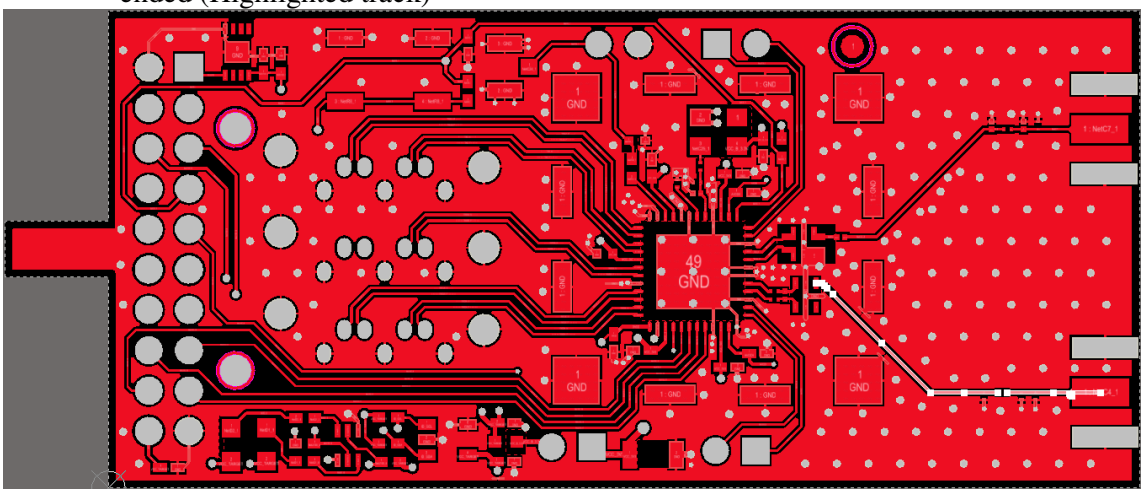
2 PCB specification

2.1 Manufacturing data

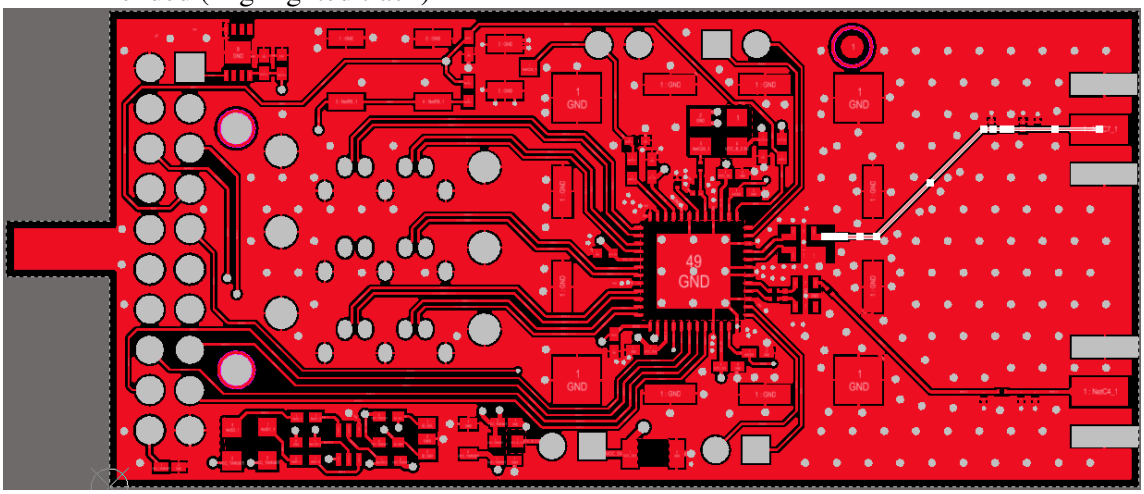
- Size: 30mm x 71.5mm
- PCB material: S1000
- Layers: 4
- Finish: ENIG
- Minimum via hole size: 0.15 mm
- Minimum via pad size: 0.35 mm
- Minimum track width: 0.1 mm (3.936mil)
- Minimum spacing: 0.15 mm (5.905mil)
- Solder mask color: Dark BLUE
- Silk-screen color: White
- Controlled impedance for LVDS tracks on top layer with 50 Ohms single ended and 100 Ohms differential (Highlighted track)



- Controlled impedance for RF tracks on top layer with 50 Ohms @ 2.4GHz single ended (Highlighted track)



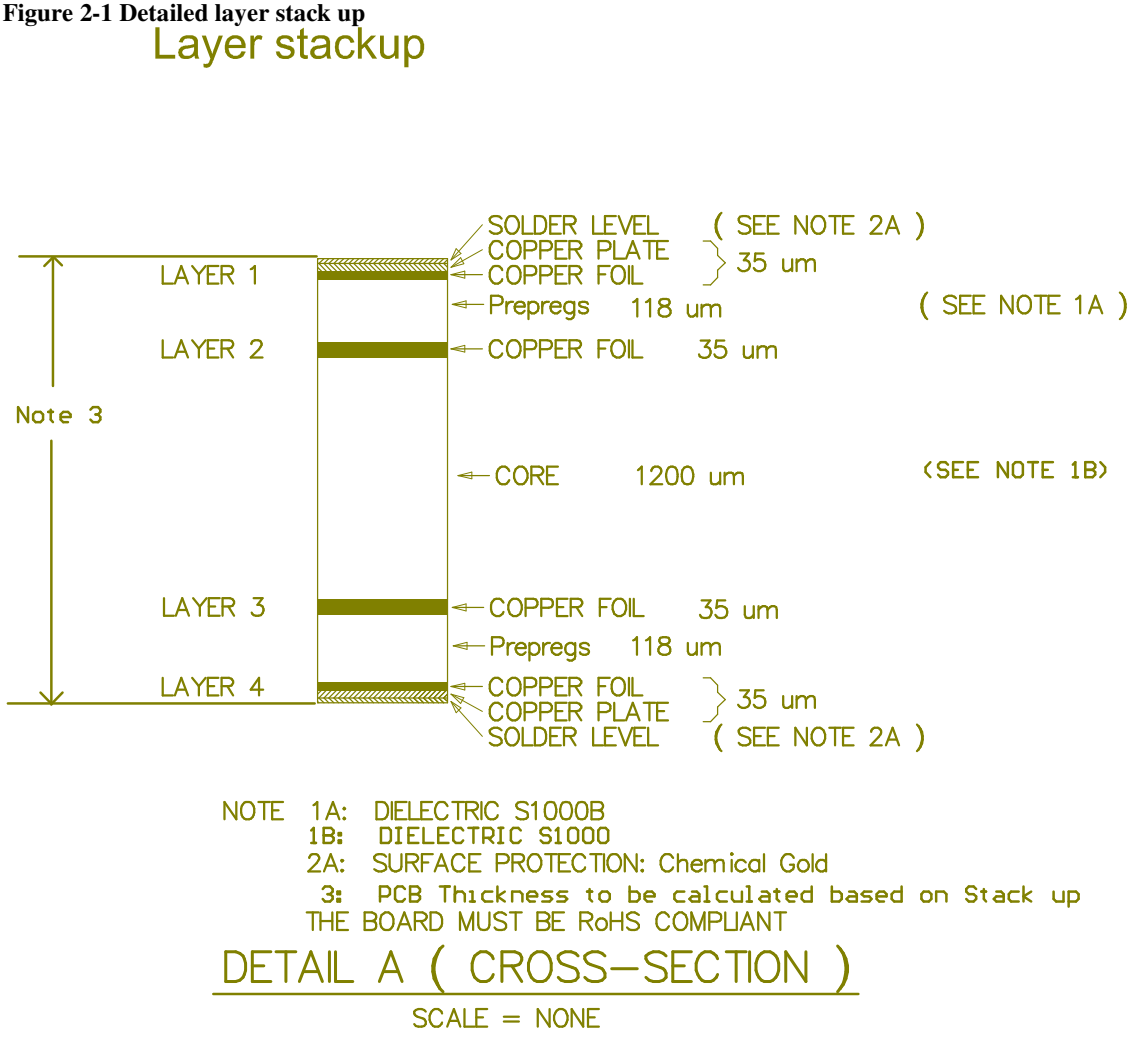
- Controlled impedance for RF tracks on top layer with 50 Ohms @ 900MHz single ended (Highlighted track)



2.2 Layer stack up

Figure Figure 2-1 shows the detailed layer stack up for this PCB.

Figure 2-1 Detailed layer stack up



2.3 Gerber files

Table 2-1 Layer stack up corresponding Gerber files (listed from top to bottom)

File name	Description
REB215_XPRO_A.GTP	Gerber file for top paste-mask
REB215_XPRO_A.GTO	Gerber file for top overlay (silkscreen)
REB215_XPRO_A.GTS	Gerber file for top solder-mask
REB215_XPRO_A.GTL	Gerber file for top layer
REB215_XPRO_A.G1	Gerber file for Mid layer 1
REB215_XPRO_A.G2	Gerber file for Mid layer 2
REB215_XPRO_A.GBL	Gerber file for bottom signal layer
REB215_XPRO_A.GBS	Gerber file for bottom solder-mask
REB215_XPRO_A.GBO	Gerber file for bottom overlay (silkscreen)

REB215_XPRO_A.GBP	Gerber file for bottom paste-mask
REB215_XPRO_A.GM1	Gerber file for mechanical 1 layer (board outline)
REB215_XPRO_A.DRR	Drill file report
REB215_XPRO_A.TXT	Drill file

2.4 Special via considerations

All vias are covered with solder mask on the top side of the board. On the bottom side vias that are used as test points have openings in the solder mask.

2.5 Placement of fabrication ID mark

The fabrication ID mark should be placed on the bottom side.

3 Panelizing

When making panels for this board the following issues should be considered.

- Fiducial marks should be placed on the panel.

4 Quality of silkscreen layers

The silkscreen layers for the PCB must be of high quality for several reasons:

- Very small text is used
- Text is close to pads and therefore the mask must be centered properly on the board
- The PCB is used for development boards and therefore the silkscreen is an essential part of the overall product quality.