

WILC3000 SD PCB Specification



Table of Contents

1	GENERAL INFORMATION.....	3
1.1	Board identification.....	3
1.2	Contact persons for PCB issues.	Error! Bookmark not defined.
2	PCB SPECIFICATION	3
2.1	Manufacturing data	3
2.2	Layer stackup	4
2.3	Gerber files.....	4
2.4	Special via considerations	5
2.5	Placement of fabrication ID mark	5
3	PANELIZING.....	5
4	QUALITY OF SILKSCREEN LAYERS	5

1 General information

1.1 Board identification

Board Name: WILC3000 SD

Board identification number: A08-2481 Rev 2

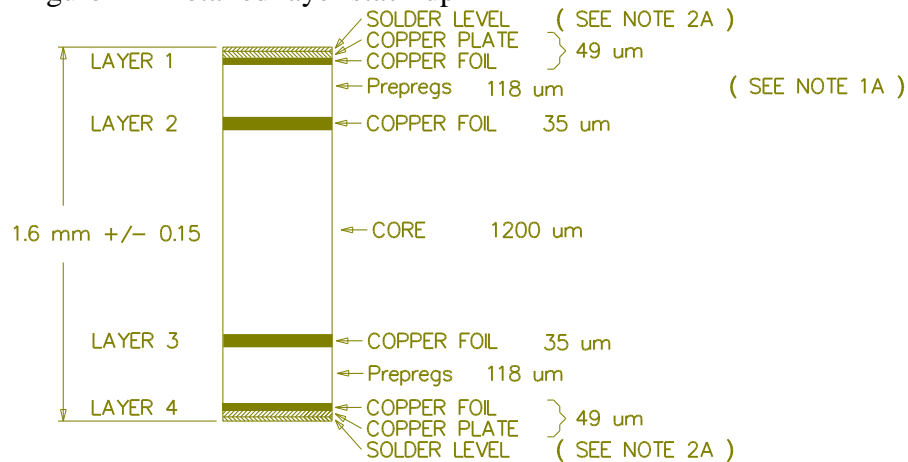
2 PCB specification

2.1 Manufacturing data

- Size: 85mm x 34mm
- PCB material: FR-4, 1.6mm thickness
- Layers: 4
- Finish: ENIG
- Minimum via hole size: 0.3 mm
- Minimum via pad size: 0.6 mm
- Minimum track width: 0.2 mm
- Minimum spacing: 0.15 mm
- Solder mask color: DARK BLUE
- Silk-screen color: White

2.2 Layer stack up

Figure 2-1 Detailed layer stack up



NOTE 1A: DIELECTRIC FR4
2A: SURFACE PROTECTION: Chemical Gold

THE BOARD MUST BE RoHS COMPLIANT

DETAIL A (CROSS-SECTION)

SCALE = NONE

2.3 Gerber files

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

File name	Description
WILC3000_SD_pcb_rev2.GTP	Gerber file for top paste-mask
WILC3000_SD_pcb_rev2.GTS	Gerber file for top solder-mask
WILC3000_SD_pcb_rev2.GTL	Gerber file for top layer
WILC3000_SD_pcb_rev2.GBL	Gerber file for bottom signal layer
WILC3000_SD_pcb_rev2.GBS	Gerber file for bottom solder-mask
WILC3000_SD_pcb_rev2.GBO	Gerber file for bottom overlay (silkscreen)
WILC3000_SD_pcb_rev2.GM2	Gerber file for mechanical 1 layer (board outline)
WILC3000_SD_pcb_rev2.GTO	Gerber file for top overlay (silkscreen)
WILC3000_SD_pcb_rev2.GP1	Gerber file for internal negative plane layer 1(GND)
WILC3000_SD_pcb_rev2.GP2	Gerber file for internal negative plane layer 2(POWER)
WILC3000_SD_pcb_rev2.DRL	NC Drill file
WILC3000_SD_pcb_rev2.DRR	Drill file Report
WILC3000_SD_SlotHoles_pcb_rev2.txt	Hole Drill File for Slot Holes
WILC3000_SD_RoundHoles_pcb_rev2.txt	Hole Drill File for Round Holes

2.4 *Special via considerations*

All plated through holes with solder mask covered area need to be filled/tented on both side of the PCB.

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.