

TITLE: PCB Specification

PAGE: 1 of 5



QT6 Xplained PRO PCB Specification

Table of Contents

1	GENERAL INFORMATION.....	3
1.1	Board identification.....	3
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: QT6 Xplained PRO

Board identification number: A08-2256 Rev 3

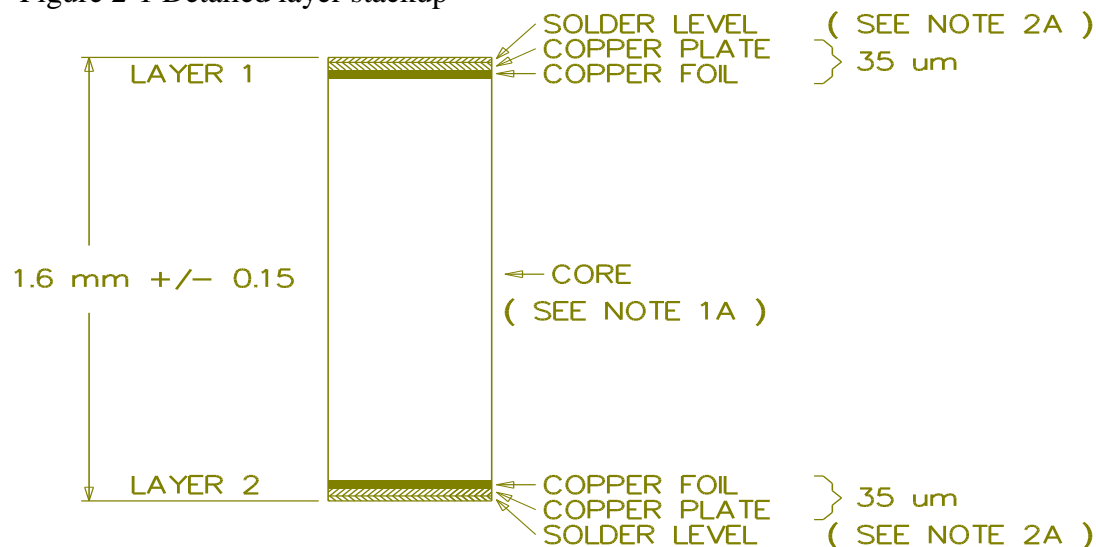
2 PCB specification

2.1 *Manufacturing data*

- Size: 85.5mm x 100mm
- PCB material: FR-4, 1.6mm thickness
- Layers: 2
- Finish: ENIG
- Minimum via hole size: 0.25 mm
- Minimum via pad size: 0.55 mm
- Minimum track width: 0.2 mm
- Minimum spacing: 0.2 mm
- Solder mask color: BLACK
- Silk-screen color: White

2.2 Layer stackup

Figure 2-1 Detailed layer stackup



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 stackup corresponding Gerber files (listed from top to bottom)

File name	Description
QT6_Xplained_Pro.GTP	Gerber file for top paste-mask
QT6_Xplained_Pro.GTS	Gerber file for top solder-mask
QT6_Xplained_Pro.GTL	Gerber file for top layer
QT6_Xplained_Pro.GBL	Gerber file for bottom signal layer
QT6_Xplained_Pro.GBS	Gerber file for bottom solder-mask
QT6_Xplained_Pro.GBO	Gerber file for bottom overlay (silkscreen)
QT6_Xplained_Pro.GBP	Gerber file for bottom paste-mask
QT6_Xplained_Pro.GM1	Gerber file for mechanical 1 layer (board outline)
QT6_Xplained_Pro.GD1	Drill Drawing
QT6_Xplained_Pro.GG1	Drill Guide
QT6_Xplained_Pro.DRL	NC Drill file
QT6_Xplained_Pro.DRR	Drill file Report
QT6_Xplained_Pro.txt	Hole Drill File

2.4 *Special via considerations*

All plated through holes with solder mask covered area need to be 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.