

# **SAMC21N Xplained Pro PCB Specification**



## Table of Contents

<b>1</b>	<b>GENERAL INFORMATION.....</b>	<b>3</b>
<b>1.1</b>	<b>Board identification.....</b>	<b>3</b>
<b>2</b>	<b>PCB SPECIFICATION .....</b>	<b>3</b>
<b>2.1</b>	<b>Manufacturing data .....</b>	<b>3</b>
<b>2.2</b>	<b>Layer stack up .....</b>	<b>4</b>
<b>2.3</b>	<b>Gerber files.....</b>	<b>5</b>
<b>2.4</b>	<b>Special via considerations .....</b>	<b>5</b>
<b>2.5</b>	<b>Placement of fabrication ID mark .....</b>	<b>5</b>
<b>2.6</b>	<b>Controlled Impedance.....</b>	<b>5</b>
<b>3</b>	<b>PANELIZING.....</b>	<b>6</b>
<b>4</b>	<b>QUALITY OF SILKSCREEN LAYERS .....</b>	<b>6</b>

## 1 General information

### 1.1 Board identification

Name: SAMC21N Xplained Pro

Board identification number: A08-2581 Rev2

## 2 PCB specification

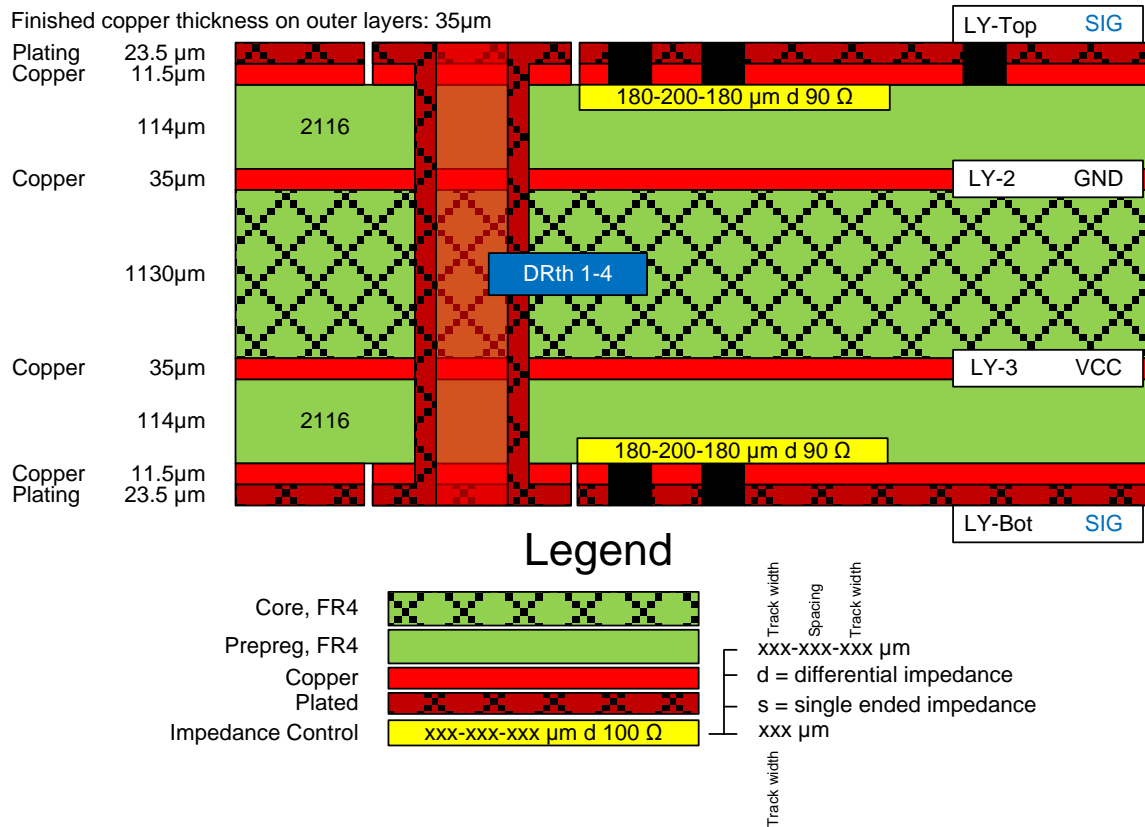
### 2.1 Manufacturing data

- Size: 60mm x 100mm
- PCB material: FR-4, 1.6mm thickness
- Layers: 4
- Finish: ENIG
- Minimum via hole size: 0.20 mm
- Minimum via pad size: 0.40 mm
- Minimum track width: 0.125 mm (4.9mil)
- Minimum spacing: 0.12 mm (4.6mil)
- Solder mask color: Dark BLUE
- Silk-screen color: White
- Controlled impedance on top side for differential USB tracks (180-220-180 um)

## 2.2 Layer stack up

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
SAMC21N_Xplained_Pro.GTP	Gerber file for top paste-mask
SAMC21N_Xplained_Pro.GTO	Gerber file for top overlay (silkscreen)
SAMC21N_Xplained_Pro.GTS	Gerber file for top solder-mask
SAMC21N_Xplained_Pro.GTL	Gerber file for top layer
SAMC21N_Xplained_Pro.GP1	Gerber file for internal negative plane layer 1 (GND layer)
SAMC21N_Xplained_Pro.GP2	Gerber file for internal negative plane layer 2 (power layer)
SAMC21N_Xplained_Pro.GBL	Gerber file for bottom signal layer
SAMC21N_Xplained_Pro.GBS	Gerber file for bottom solder-mask
SAMC21N_Xplained_Pro.GBO	Gerber file for bottom overlay (silkscreen)
SAMC21N_Xplained_Pro.GBP	Gerber file for bottom paste-mask
SAMC21N_Xplained_Pro.GM1	Gerber file for mechanical 1 layer (board outline)
SAMC21N_Xplained_Pro.DRR	NC Drill report file
SAMC21N_Xplained_Pro-RoundHoles.txt	Text file for Round holes
SAMC21N_Xplained_Pro-SlotHoles.txt	Text file for Slot holes

## 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.

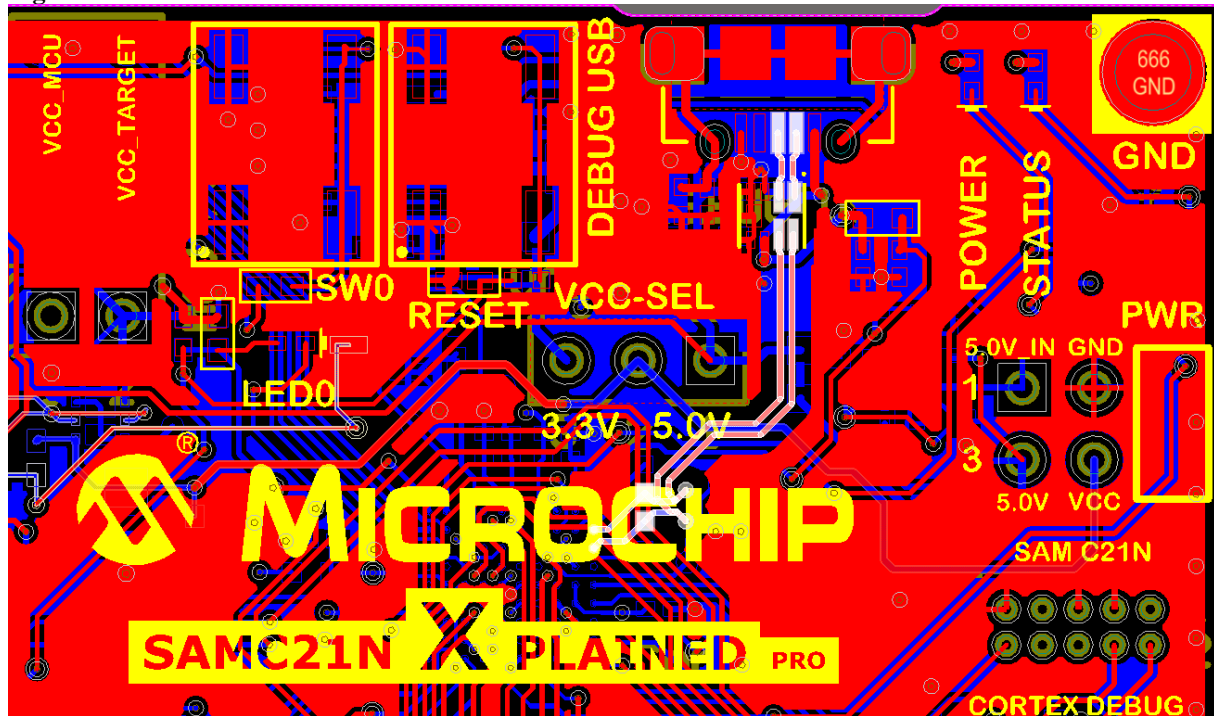
## 2.5 Placement of fabrication ID mark

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

## 2.6 Controlled Impedance

Controlled impedance for USB tracks on top and bottom layer with 90 Ohms differential.

Figure 2-2 USB tracks



### 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.