

88CKSCKT-XPRO PCB Specification

88CKSCKTSOIC-XPRO
88CKSCKTUDFN-XPRO
88CKSCKTRBH-XPRO

Original Date: Apr 26th, 2016

Table of Contents

1	GENERAL INFORMATION.....	3
1.1	Board identification.....	3
1.2	Contact persons for PCB issues.	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.....	4
2.5	Placement of fabrication ID mark	4
3	PANELIZING.....	5
4	QUALITY OF SILKSCREEN LAYERS	5

1 General information

1.1 Board identification

Name: 88CKSCKT-XPRO

Board identification number: A08-2550.

This PCB is planned for use on three kits.

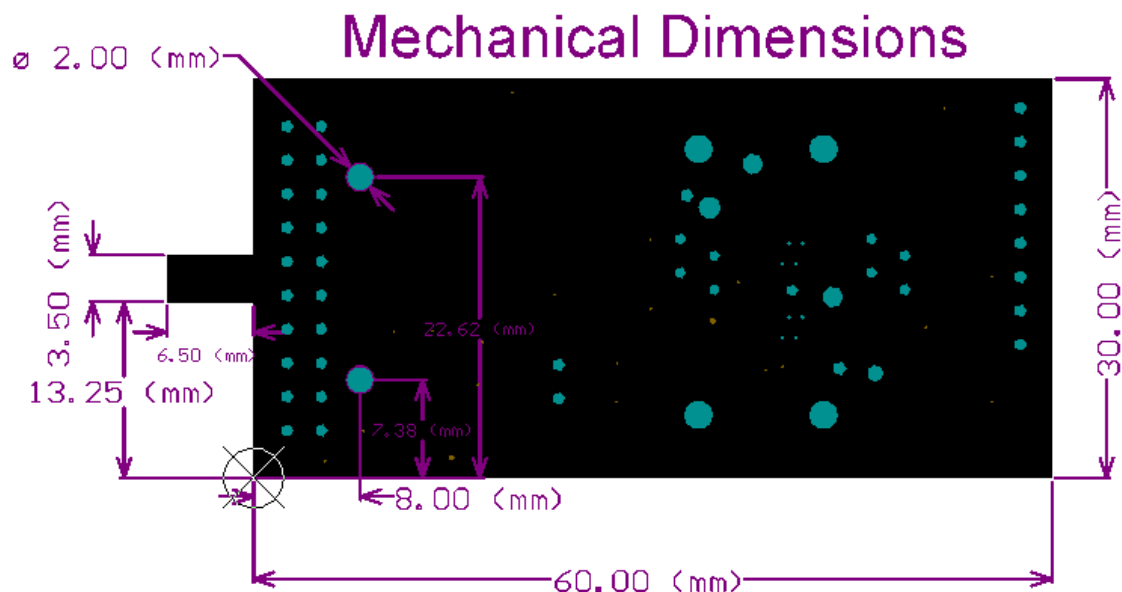
1.2 Contact persons for PCB issues.

- PCB Designer/Engineer: Jim Boomer, james.boomer@atmel.com, 719-540-1528

2 PCB specification

2.1 Manufacturing data

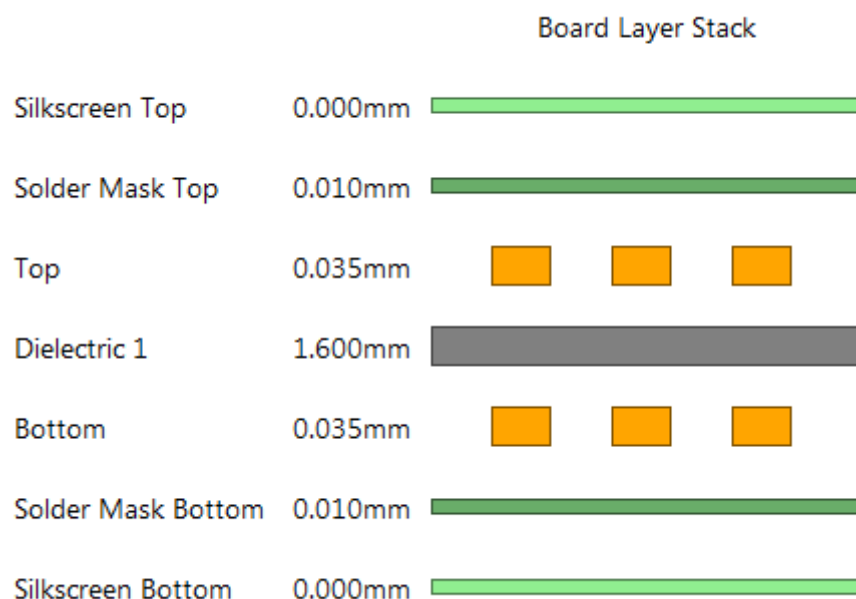
- Size: 30mm x 60mm
- PCB material: FR370HR, 1.6mm thickness
- Layers: 2
- Finish: ENIG
- Minimum via hole size: 0.25mm (9.84mil)
- Minimum via pad size: 0.55mm (21.65mil)
- Minimum track width: 0.203mm (8mil)
- Minimum spacing: 0.175mm (7mil)
- Solder mask color: Dark Red
- Silk-screen color: White (Both Sides)



2.2 Layer stackup

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

Figure 2-1 Detailed layer stackup



2.3 Gerber files

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

File name	Description
88CKSCKT_XPRO.GTP	Gerber file for top paste-mask
88CKSCKT_XPRO.GTO	Gerber file for top overlay (silkscreen)
88CKSCKT_XPRO.GTS	Gerber file for top solder-mask
88CKSCKT_XPRO.GTL	Gerber file for top layer
88CKSCKT_XPRO.GBL	Gerber file for bottom signal layer
88CKSCKT_XPRO.GBS	Gerber file for bottom solder-mask
88CKSCKT_XPRO.GBO	Gerber file for bottom overlay (silkscreen)
88CKSCKT_XPRO.GBP	Gerber file for bottom paste-mask
88CKSCKT_XPRO.GM1	Gerber file for mechanical 1 layer (board outline)
88CKSCKT_XPRO.DRR	Drill file report
88CKSCKT_XPRO.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.

- 3 Fiducials do exist on each board on the top side where components are placed.
- Fiducial marks may be placed on the panel.

4 Quality of silkscreen layers

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

- Smallest dimensions used for the silkscreen:
 - 0.524mm Height. 0.115Width
 - Location: Text Back side of the PCB near XPRO Connector Pins
- 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.