

# CryptoAuth-XPRO-B Specification

Revision Date: May 13th, 2017

## Table of Contents

<b>1</b>	<b>GENERAL INFORMATION.....</b>	<b>3</b>
1.1	Board identification.....	3
1.2	Contact persons for PCB issues. ....	3
<b>2</b>	<b>PCB SPECIFICATION .....</b>	<b>3</b>
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
<b>3</b>	<b>PANELIZING.....</b>	<b>5</b>
<b>4</b>	<b>QUALITY OF SILKSCREEN LAYERS .....</b>	<b>5</b>

## 1 General information

### 1.1 Board identification

Name: CryptoAuth Xplained Pro

Board identification number: A08-2825

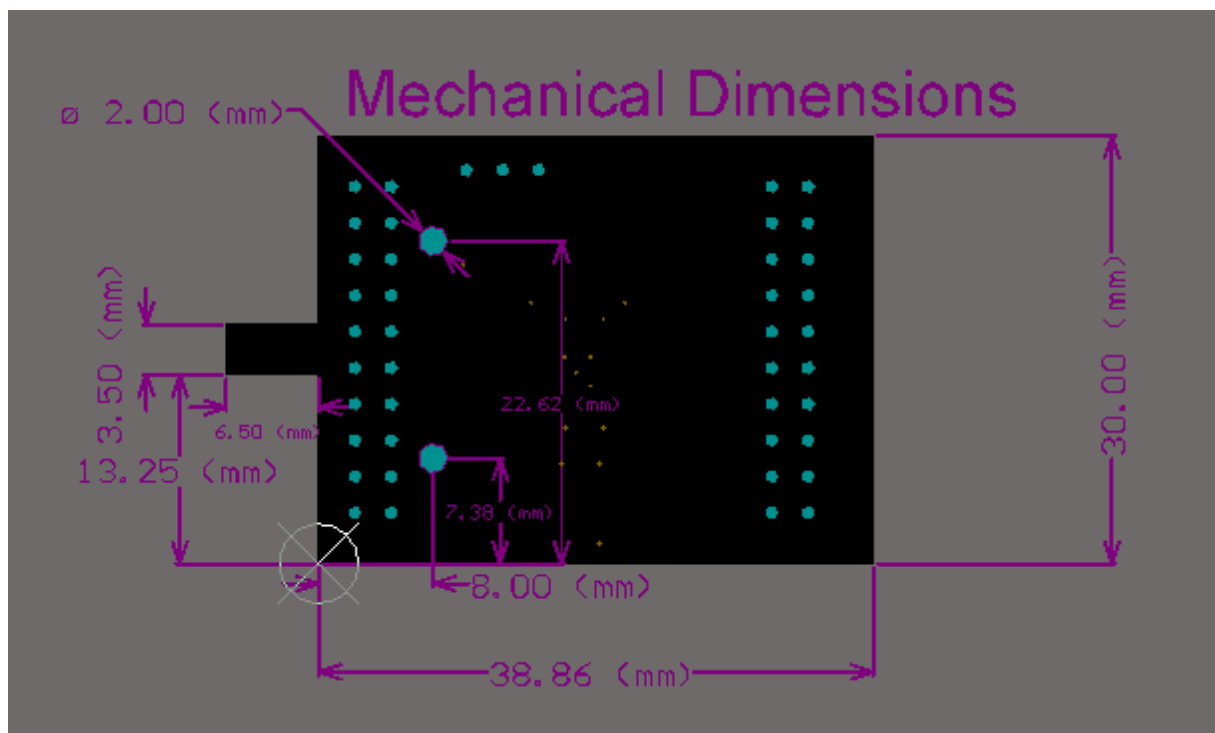
### 1.2 Contact persons for PCB issues.

- PCB Designer: Jim Boomer, [james.boomer@microchip.com](mailto:james.boomer@microchip.com), 719-540-1528

## 2 PCB specification

### 2.1 Manufacturing data

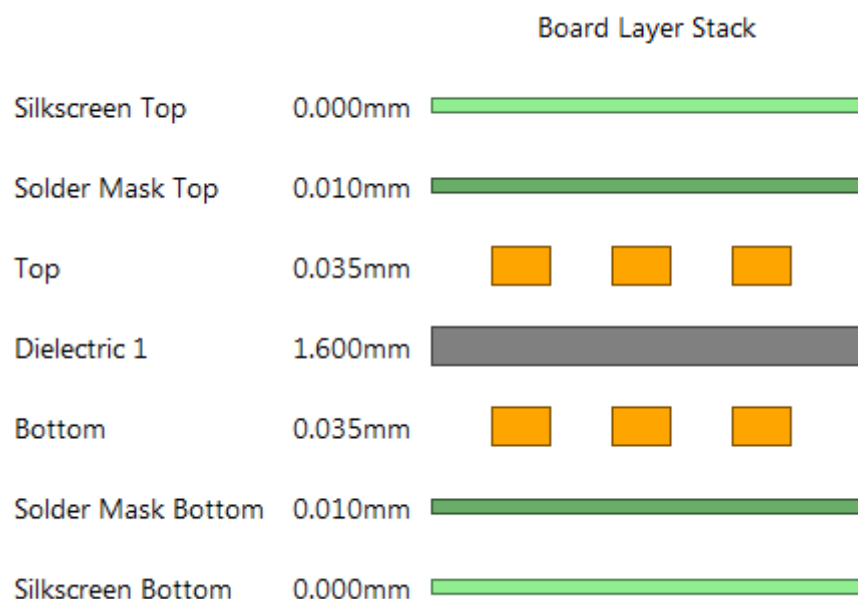
- Size: 30mm x 45.36mm
- PCB material: FR370HR, 1.6mm thickness
- Layers: 2
- Finish: ENIG
- Minimum via hole size: 0.381mm (15mil)
- Minimum via pad size: 0.762mm (30mil)
- Minimum track width: 0.175mm (7mil)
- Minimum spacing: 0.15mm (6mil)
- Solder mask color: Dark Red
- Silk-screen color: White



## 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
CryptoAuth-XPRO-B.GTP	Gerber file for top paste-mask
CryptoAuth-XPRO-B.GTO	Gerber file for top overlay (silkscreen)
CryptoAuth-XPRO-B.GTS	Gerber file for top solder-mask
CryptoAuth-XPRO-B.GTL	Gerber file for top layer
CryptoAuth-XPRO-B.GBL	Gerber file for bottom signal layer
CryptoAuth-XPRO-B.GBS	Gerber file for bottom solder-mask
CryptoAuth-XPRO-B.GBO	Gerber file for bottom overlay (silkscreen)
CryptoAuth-XPRO-B.GBP	Gerber file for bottom paste-mask
CryptoAuth-XPRO-B.GM1	Gerber file for mechanical 1 layer (board outline)
CryptoAuth-XPRO-B.DRR	Drill file report
CryptoAuth-XPRO-B.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:

- Smallest dimensions used for the silkscreen:
  - 5 mils (.127mm)
  - Location: Back side of the PCB
- 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.