

## \*\*\* PCB SPECIFICATION FOR BARE BOARD MANUFACTURING \*\*\*

PRODUCT OWNER : Silicon Labs  
DOCUMENT/BOARD : PCB4207A

DATE : 2020. 07. 15.  
REVISION : A00

PREPARED BY : Tamas Bodi  
BOARDS pr PANEL : 12 (4 x 3)  
PANEL SIZE : 210.2 x 205.4 mm  
BOARD SIZE : 30.0 x 60.0 mm  
BOARD THICKNESS : 1.6 mm +/-10%  
NO OF LAYERS : 4  
MATERIAL(S) : Glass Epoxy FR-4, NEMA Class 2, UL 94V-0, Tg min 150 C  
Materials in compliance with the RoHS and WEEE directives  
MARKINGS : Logo, Week/Year, UL (ON SECONDARY SIDE (BOT))  
(Avoid areas reserved for DataMatrix, Barcodes or Labels)  
All PCB manufacturer's markings (Logo, Week/Year, UL)  
shall be put in the PCB frame. No marking on the boards  
is allowed  
QUALITY REQ. : IPC-A-600 (current revisions) Class 2, and IPC specifications  
referred to by IPC-A-600  
GENERAL REQ. : - Copper must not be added or removed from inside the board  
outline(s), without written consent/approval.  
Use the balancing of the panel that comes with the  
Gerber files (without alterations)  
If applicable, the following requirements are valid:  
- If Build-Up (Stack-Up) is specified, follow Build-Up,  
otherwise use (board manufacturer) standard Build-Up.  
- Break-away areas may be used for patterns, holes etc.  
by manufacturer for QA purposes.  
- If V-CUT, use angle 30 +/- 5 degrees.  
V-CUT minimum remaining thickness 0.5 +/- 0.1 mm.  
Use of V-CUT test pads is allowed.  
- Inner radius (contour/outline) 1.2 mm, unless stated  
otherwise.  
COPPER THK. : SEE BUILD-UP  
COPPER PASSIV. : ENIG to meet IPC-4552 requirements (current revision)  
(Electroless Nickel/Immersion Gold)  
RESIST MASK : Solder Mask Color: BLACK (NB! NON-STANDARD)  
Photo Polymer Wet film  
to IPC-SM-840 Class T requirements (current revision)  
Thickness minimum 8 um, maximum 20 um  
Removal of thin solder resist dams (between QFN and BGA pads)  
is allowed.  
VIA HOLES : PLUGGED/FILLED, IPC-4761 (current revision) Type IV-b  
Plugged and Covered Both Sides, Low CTE Plugging Paste  
If Type IV-b is not available as a process, then Type IV-a  
for the Top Side, and Overprinted (Tented) Bot Side is OK  
LEGEND/SILKSCR. : WHITE, BOTH SIDES (TOP + BOT)  
NOMINAL VALUES for Width, Spacing and VIA Diameter:  
Cu TRACK(TRACE) : Minimum conductor width : 0.10 mm (4 mils)  
Cu TRACK(TRACE) : Minimum conductor spacing : 0.10 mm (4 mils)  
MINIMUM VIA : Minimum via pad diameter : 0.50 mm (20 mils)  
Minimum via hole diameter : 0.25 mm (9.8 mils)  
Min via hole may have more than one pad diameter.

(SPECIFICATION CONTINUED ON NEXT PAGE)

## BUILD UP

```

:
TOP ==|||===== 35 um Cu (ca) After plating
  ////| | PREPREG ////////////// 132 um 2116 *
L1 ==|||===== 18 um Cu (0.5 Oz)
  ////| | CORE ////////////// 1156 um **
L2 ==|||===== 18 um Cu (0.5 Oz)
  - - -| | PREPREG ////////////// 132 um 2116 *
BOT ==|||===== 35 um Cu (ca) After plating

```

\*)

The distance between Top-L1 and Bottom-L2 should be as close to 132 um as possible!

\*\*)

Select Center Core thickness in order to reach specified board thickness

## TEST

```

: 100% Electrical Test
  Optical test, AOI (with automatic scanner)
  Visual inspection
  (Generate netlist from Gerber and Drill files)

```

Avoid use of 2125 Prepreg

If NB! is used in this specification, it means:  
 abbreviation for nota bene!, a Latin expression meaning  
 note well!

## NC DRILL - HOLE INFORMATION:

WARNING : Drill dimensions must be taken from the Excellon (.DRL) file(s), and the drill report file(s) (.DRR).  
 NON-PLATED holes may have a small center marker in the Gerber files.  
 Under no circumstance must these Gerber flashes be mistaken for the hole drill dimensions!

The drill data may contain slots (in a separate file).

Dimensions for the finished board (after plating).

Tolerances +/- 0.1 mm, unless specified differently.

Via Holes +0.075 mm/-Via Size, unless specified differently.