

*** PCB SPECIFICATION FOR BARE BOARD MANUFACTURING ***

PRODUCT OWNER : Silicon Labs
DOCUMENT/BOARD : PCB4205A

DATE : 2020. 10. 14.
REVISION : A00

PREPARED BY : Adam Sule
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: MATTE BLACK
Photo Polymer Wet film
to IPC-SM-840 Class T requirements (current revision)
Thickness minimum 8 um, maximum 40 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

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.