

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

PRODUCT OWNER : SiLabs
DOCUMENT/BOARD : PCB2010A Rev C02
DATE : 2019-10-09
REVISION : C02

PREPARED BY : Ole Jacob Bryhni Frostad
BOARDS pr PANEL: 4 (1 x 4)
PANEL SIZE : 131.0 x 280.0 mm
BOARD SIZE : 115.0 x 60.0 mm
BOARD THICKNESS: 1.6 mm +/- 10 %
NO OF LAYERS : 6
MATERIAL(S) : Glass Epoxy FR-4, IPC-4101 (current revision) /99 or /124 (Tg min 150 C)
MATERIAL(S) : Glass Epoxy FR-4, IPC-4101 (current revision) /126 or /129
Td min 340 C (Decomposition Temperature), Tg min 170 C
CTE, Z-axis max 3.5 % (50 - 260 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 Lables
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.
If applicable, the following requirements are valid:
- Copper balancing may be applied on break-away-tabs,
or otherwise outside board outline(s), but must have
a minimum 1.5 mm clearance to possible fiducials.
- 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 Class 2 requirements (current revision)
(Electroless Nickel/Immersion Gold)
RESIST MASK : Solder Mask Color: BLACK (NB! NON-STANDARD)
Photo Polymer Wet film
Thickness minimum 8 um, maximum 20 um
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)
CONTROLLED IMP : Design has Controlled impedances. FOLLOW BUILD-UP STRICTLY!
NOMINAL VALUES for Width, Spacing and VIA Diameter:
Cu TRACK(TRACE): Minimum conductor width : 0.10 mm
Cu SPACING : Minimum conductor spacing: 0.10 mm
MINIMUM VIA : Minimum via pad diameter : 0.51 mm
Min via hole may have more than one pad diameter.

(SPECIFICATION CONTINUED ON NEXT PAGE)

BUILD-UP :

```
L1  =====| |===== 35 um Cu (ca) After plating
      /////| |///// C O R E ///// 300 um
L2  =====| |===== 18 um Cu (0.5 Oz)
      - - -| |- P R E P R E G - - - -
L3  =====| |===== 18 um Cu
      /////| |///// C O R E ///// - - CENTER - -
L4  =====| |===== 18 um Cu
      - - -| |- P R E P R E G - - - -
L5  =====| |===== 18 um Cu
      /////| |///// C O R E ///// 300 um
L6  =====| |===== 35 um Cu
```

Select other thicknesses to reach specified board thickness
(Foil Build instead of Core Build is acceptable)

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 is latin,
meaning "mark well" or "observe particularly"

Nominal tolerances (if no other tolerances given)
PTH +/- 0.10 mm for d <= 2.0 mm
PTH +/- 0.15 mm for 2.0 < d <= 5.3 mm
PTH +/- 0.20 mm for d > 5.3 mm
NPTH +/- 0.05 mm for d <= 5.3 mm
NPTH +/- 0.10 mm for d > 5.3 mm

Contec Electronics AS * ECAD Center * Norway
email: post@contecel.com URL: www.contecel.com Phone: +47 6677 5340

+++ YOUR CIRCUIT BOARD DESIGN PARTNER +++
----->