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

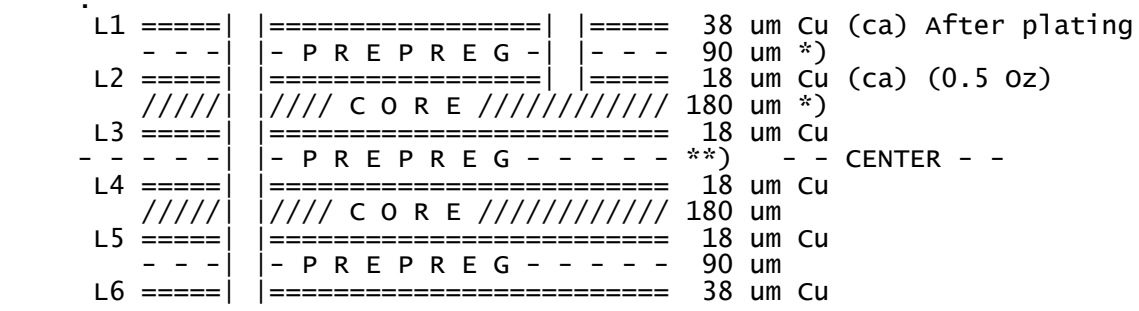
PRODUCT OWNER : Silicon Labs
DOCUMENT/BOARD : PCB4103A

DATE : 2016.11.16.
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 51 mm
BOARD THICKNESS: 1.6 mm +/- 10 %
NO OF LAYERS : 6
MATERIAL(S) : Glass Epoxy FR-4, IPC-4101 (current revision) /124, /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)
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
refered 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.
- 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 2 (T) requirements (current revision)
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
u-Vias (microvias) must be 100% Copper Filled
LEGEND/SILKSCR.: WHITE, BOTH SIDES (TOP + BOT)
CONTROLLED IMP : Design has Controlled impedances. FOLLOW BUILD-UP STRICTLY!
Unless explicitly stated otherwise, controlled impedance
has been designed into the board. Use of test strip is
hence normally not required.
NOMINAL VALUES for Width, Spacing and VIA Diameter:
Cu TRACK(TRACE): Minimum conductor width : 0.10 mm (4 mils)
Cu SPACING : Minimum conductor spacing : 0.10 mm (4 mils)
MINIMUM VIA : Minimum via pad diameter : 0.33 mm (13 mils) u-vias
Minimum via hole diameter : 0.15 mm (5.9 mils) u-vias
Min via hole may have more than one pad diameter.
MINIMUM VIA : Minimum via pad diameter : 0.51 mm (20 mils) Normal vias
Minimum via hole diameter : 0.25 mm (9.8 mils) Normal vias
Min via hole may have more than one pad diameter.
Blind vias L1-L2
u-vias L1-L2

(SPECIFICATION CONTINUED ON NEXT PAGE)

BUILD UP :



*)
The distance from bottom of L1 to top of L3 should be
as close to 300 um as possible!

**)
Select Center Prepreg 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!"
