

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

PRODUCT OWNER : Silicon Labs  
DOCUMENT/BOARD : PCB4502 Rev B01  
DATE : 2015-03-24  
REVISION : B01

PREPARED BY : Ole Jacob Bryhni Frostad  
BOARDS pr PANEL: 12 (4 x 3)  
PANEL SIZE : 210.2 x 205.4 mm  
BOARD SIZE : 30.0 x 41.0 mm  
BOARD THICKNESS: 1.6 mm +/- 10 %  
NO OF LAYERS : 6  
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 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 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  
VIA HOLES : PLUGGED/FILLED, Class 2, IPC-4761 (current revision) Type IV-b  
Plugged and Covered Both Sides, Low CTE Plugging Paste  
THESE BOARDS MUST NOT BE PLUGGED  
LEGEND/SILKSCR.: WHITE, BOTH SIDES (TOP + BOT)  
Separate Silk Top/Bot for PCB4502A, PCB4502B, PCB4502C and PCB4502D  
All other PCB files are equal for the PCB4502 boards A, B, C and D  
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: 0.0889 mm (3.5 mils)  
MINIMUM VIA : Minimum via pad diameter : 0.51 mm (20 mils)  
Min via hole (SEE HOLE INFORMATION FURTHER DOWN)  
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 /////////// 304 um
L2 ===== 18 um Cu (0.5 Oz)
  - - - - P R E P R E G - - - - 100 um
L3 ===== 18 um Cu
  - - /////////// C O R E /////////// 700 um - - CENTER - -
L4 ===== 18 um Cu
  - - - - P R E P R E G - - - - 100 um
L5 ===== 18 um Cu
  /////////// C O R E /////////// 304 um
L6 ===== 35 um Cu

```

(Approximate Prepreg thicknesses)

Prepreg thickness may be adjusted in order  
to reach target 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 (.exc) file(s).  
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 file may contain slots. See drill information below.  
The Gerber file mb4502.gex may also contain slot information.  
Dimensions for the finished board (after plating).  
Tolerances +/- 0.1 mm, unless specified otherwise below.  
Via Holes +0.05 mm/-Via Size, unless specified otherwise below.

#### PLATED HOLES:

```

T01 VH  DIA = 0.25 mm  QTY = 2076 (VIA-HOLES)
T02 VH  DIA = 0.3 mm   QTY = 1728 (VIA-HOLES)
T03 PTH DIA = 1.5 mm   QTY = 12
T04 PTH DIA = 1.6 mm   QTY = 48

```

#### NON-PLATED HOLES:

```

T05 NP  DIA = 1.35 mm  QTY = 48
T06 NP  DIA = 3.0 mm   QTY = 2

```

Contec Electronics AS \* ECAD Center \* Norway  
email: post@contecel.com URL: www.contecel.com  
Phone: +47 6677 5340 or +47 472 55 462 Fax: +47 6677 5341

+++ YOUR CIRCUIT BOARD DESIGN PARTNER +++