

*** PRODUCT SPECIFICATION FOR CIRCUIT BOARD MANUFACTURING ***

PRODUCT OWNER : Energy Micro AS
DOCUMENT/BOARD : PCB3500 REV.A00
DATE : 2011-07-13
REVISION : A00

PREPARED BY : Tor Finstad
BOARDS pr PANEL: 4
PANEL SIZE : 199.2 x 232.0 mm
BOARD SIZE : 88.0 x 104.0 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 140 C
Materials in compliance with the RoHS and WEEE directives
MARKINGS: Logo, Week/Year, UL
QUALITY REQ. : IPC-A-600 Class 2, IPC-6010 Series (current revisions),
and all other relevant IPC specifications.
GENERAL REQ. : If applicable, the following requirements are valid:
- If Build-Up (Stack-Up) is specified, follow Build-Up,
otherwise use (board manufacturer) standard Build-Up.
- 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.
- Break-away areas may be used for patterns, holes etc
by manufacturer for QA purposes.
- Inner radius (contour) 1.2 mm, unless stated otherwise.
- Manufacturer may plug tented via-holes to improve yield.
COPPER THK. : Cu-thickness outer layers: 35 um Finished Board
COPPER (INNER) : Cu-thickness inner layers : 35 um
COPPER PASSIV. : ENIG to meet IPC-4552 requirements
(Electroless Nickel/Immersion Gold)
UNLESS OPTIONALLY: EXPLICIT OTHER PASSIVATION REQUESTED
RESIST MASK : Photo Polymer Wet film, BLACK NB!
to IPC-SM-840 Class T requirements (current revision)
VIA HOLES : TENTED (OVERPRINTED, NOT PLUGGED)
LEGEND COLOUR : Silk screen colour: WHITE
LEGEND LAYER(S): Silk screen legend: BOTH SIDES
NOMINAL VALUES for Width, Spacing and VIA Diameter:
Cu TRACK(TRACE): Minimum conductor width : 0.127 mm (5 mils)
Cu SPACING : Minimum conductor spacing: 0.127 mm (5 mils)
MINIMUM VIA : Minimum via-pad diameter : 0.66 mm (26 mils)
Min via hole (SEE HOLE INFORMATION FURTHER DOWN)
Min via hole may have more than one pad diameter.
TEST : 100% Electrical Test
Optical test, AOI (with automatic scanner)
Visual inspection
(Generate netlist from Gerber/Drill files)

(SPECIFICATION CONTINUED ON NEXT PAGE)

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!

Dimensions for the finished board (after plating).
Tolerances +/- 0.1 mm, unless specified otherwise below.
Via Holes +0.1 mm/-Via Size, unless specified otherwise below.

PLATED HOLES:

T01 VH DIA = 0.3 mm QTY = 1616 (VIA-HOLES)
T02 PTH DIA = 0.4 mm QTY = 1432
T03 PTH DIA = 1.0 mm QTY = 1120
T04 PTH DIA = 3.2 mm QTY = 8

NON-PLATED HOLES:

T05 NP DIA = 1.2 mm QTY = 16

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 +++

----->