

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

PRODUCT OWNER : Energy Micro AS
DOCUMENT/BOARD : PCB3302 REV.A00
DATE : 2011-10-26
REVISION : A00

PREPARED BY : Ole Jacob Bryhni Frostad
BOARDS pr PANEL: 9 (3 x 3)
PANEL SIZE : Single Board (or panel if specified with the order)
PANEL SIZE : 205.4 x 244.4 mm
BOARD SIZE : 55.0 x 64.0 mm
BOARD THICKNESS: 1.6 mm +/- 10 %
NO OF LAYERS : 8
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
(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. : 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 (1.0 Oz)
COPPER (INNER) : Cu thickness inner layers : 18 um (0.5 Oz)
COPPER PASSIV. : ENIG to meet IPC-4552 requirements, except Nickel thickness
must be minimum 4 um. (Electroless Nickel/Immersion Gold)
RESIST MASK : Photo Polymer Wet film, BLACK NB!
to IPC-SM-840 Class T requirements (current revision)
VIA HOLES : TENTED (OVERPRINTED, NOT PLUGGED)
UNLESS OPTIONALLY: EXPLICIT OTHER VIA TREATMENT REQUESTED
LEGEND COLOUR : WHITE
LEGEND LAYER(S) : BOTH SIDES
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 : 0.48 mm (19 mils)
Min via hole (SEE HOLE INFORMATION FURTHER DOWN)
Min via hole may have more than one pad diameter.

(SPECIFICATION CONTINUED ON NEXT PAGE)

TEST : 100% Electrical Test
Optical test, AOI (with automatic scanner)
Visual inspection
(Generate netlist from Gerber/Drill files)

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 = 0.25 mm QTY = 5895 (VIA-HOLES)
T02 VH DIA = 0.3 mm QTY = 702 (VIA-HOLES)
T03 PTH DIA = 0.8 mm QTY = 432

NON-PLATED HOLES:

T04 NP DIA = 1.2 mm QTY = 36

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 +++
----->