

<-----  
 \*\*\* PCB SPECIFICATION FOR CIRCUIT BOARD MANUFACTURING \*\*\*  
 -----

PRODUCT OWNER : Energy Micro AS  
 DOCUMENT/BOARD : PCB3800 REV. A00  
 DATE : 2012-12-12  
 REVISION : A00

-----  
 PREPARED BY : Ole Jacob Bryhni Frostad  
 BOARDS pr PANEL: 9 (3 x 3)  
 PANEL SIZE : 205.4 x 244.4 mm  
 BOARD SIZE : 55.0 x 68.0 mm  
 BOARD THICKNESS: 1.6 mm +/- 10 %  
 NO OF LAYERS : 8  
 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  
                   (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. : SEE BUILD-UP  
 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)  
                   Thickness minimum 8 um, maximum 20 um  
 VIA HOLES : TENTED (OVERPRINTED, NOT PLUGGED) IPC-4761 Type I  
                   UNLESS OPTIONALLY: EXPLICIT OTHER VIA TREATMENT REQUESTED  
 LEGEND COLOUR : WHITE  
 LEGEND LAYER(S) : BOTH SIDES  
 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 : 0.48 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)

BUILD UP :

```

L1 ===== 35 um Cu (1.0 Oz)
- - - - P R E P R E G - - - - 199 um
L2 ===== 18 um Cu (0.5 Oz)
////////// C O R E ////////// 200 um
L3 ===== 18 um Cu
- - - - P R E P R E G - - - - 202 um
L4 ===== 18 um Cu
- - ////////// C O R E ////////// 200 um - - CENTER - -
L5 ===== 18 um Cu
- - - - P R E P R E G - - - - 202 um
L6 ===== 18 um Cu
////////// C O R E ////////// 200 um
L7 ===== 18 um Cu
- - - - P R E P R E G - - - - 199 um
L8 ===== 35 um Cu
    
```

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

The drill file may contain slots. See drill information below.  
 The Gerber file mb3800.gex may also contain slot information.  
 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 = 5094 (VIA-HOLES)  
 T02 VH DIA = 0.3 mm QTY = 1782 (VIA-HOLES)

NON-PLATED HOLES:  
 T03 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 +++