APPLICATION		REVISIONS				
NEXT ASSY	USED ON	SYM	DESCRIPTION	CN	DATE	APVD
NUMEROUS	NUMEROUS		RELEASED PER A/R 4192.	-----	07/31/12	S. MYERS
		A	REVISED SHEET 4 AS MARKED.	96928	11/07/13	S. MYERS

WRITER J. LANGE / S. MYERS	07/20/12	CARLETON Life Support Systems
CHK S. MYERS	07/31/12	
ENGR L. ROGNESS	07/26/12	
APPROVED MANAGERS/DIRECTORS	07/31/12	PRINTED CIRCUIT BOARDS, MANUFACTURING AND STORAGE STANDARD FOR
CONTRACT NO.		
APPROVAL		

SIZE	CAGE CODE	
A	**99251**	**ST1637822**
REVISION LEVEL **A**		SHEET **1 OF 8**

1. PURPOSE AND SCOPE

The purpose of this Manufacturing Standard is to define the minimum requirements and acceptable quality for printed circuit boards manufactured by or for CLSS when the engineering drawing or applicable document(s) do not otherwise indicate specific requirements. In cases where the engineering drawing deviates from the requirements listed herein, the engineering drawing shall prevail. In cases where the procuring document deviates from the requirements listed herein or the engineering drawing, the procuring document shall prevail. In case of uncertain or conflicting requirements, the Engineering Department of CLSS shall be consulted for interpretation or drawing revision before proceeding with any manufacture. In cases where this standard is more restrictive than the applicable government or military specifications, the requirements of this document shall prevail. CLSS, as used herein, refers to Carleton Life Support Systems Inc.

2. APPLICABLE DOCUMENTS

The following documents form a part of this standard to the extent specified herein. Unless otherwise indicated, the issue in effect on date of invitation for bids or request for proposals shall apply. When a specific document issue is indicated below, references to the document in subsequent sections of this standard shall consist of the basic document number without repetition of the pertinent issue designations.

2.1 Government documents. There are Government documents.

2.2 Non-Government documents.

STANDARDS

Joint Industry Standards

IPC-1601	Printed Board Handling and Storage Guidelines
IPC-2221	Generic Standard on Printed Board Design
IPC-6011	General Performance Specification for Printed Boards
IPC-6012	Qualification and Performance Specification for Rigid Printed Boards
IPC-6013	Qualification and Performance Specification for Flexible Printed Boards
IPC-4101	Specification for Base Materials for Rigid and Multilayer Printed Boards
IPC-4202	Flexible Base Dielectrics for Use in Flexible Printed Circuitry
IPC-4203	Adhesive Coated Dielectric Films for Use as Cover Sheets for Flexible Printed Circuitry and Flexible Adhesive Bonding Films
IPC-4204	Flexible Metal-Clad Dielectrics for Use in Fabrication of Flexible Printed Circuitry

IPC-4562	Metal Foil for Printed Board Applications
IPC-4781	Qualification and Performance Specification of Permanent, Semi-Permanent and Temporary Legend and/or Marking Inks
IPC-SM-840	Qualification and Performance Specification of Permanent Solder Mask
IPC-TM-650	Test Methods Manual
J-STD-006	Requirements for Electronic Grade Solder Alloys and Fluxed and Non-Fluxed Solid Solders for Electronic Soldering Applications
J-STD-033	Standard for Handling, Packing, Shipping and Use of Moisture/Reflow Sensitive Surface Mount Devices

OTHER PUBLICATIONS

CLSS

PD1635353	Epoxy Application Procedure for AT Rigid to Flex Transition

3. DEFINITIONS AND ABBREVIATIONS

3.1 Only generally accepted terminology is used.

3.2 The following abbreviations apply:

CLSS	Carleton Life Support Systems Inc.
HIC	Humidity Indicator Card
RH	Relative Humidity

4. GENERAL REQUIREMENTS OR STATEMENTS

4.1 Construction. Section 5 specifies the minimum requirements for printed board construction. It is understood that the requirements stated herein will not meet all intended applications. Therefore, deviations and/or additions to this document shall be specified on the referenced engineering drawing.

4.2 Moisture. Printed boards and some electronic components are sensitive to moisture. Moisture sensitive components readily absorb moisture from their surrounding environment (e.g. ambient humidity). Absorbed moisture produces significantly high vapor pressures when exposed to extreme elevated temperatures such as solder reflow. Under certain conditions, this pressure can cause internal delamination of the materials/layers and fractures/separation of plated thru-holes. The most likely environment during which a printed board is subjected to severe enough temperature environments is during the soldering process. Both hand soldering and reflow soldering can cause sufficient vapor pressure for damage. Therefore, proper precautions must be taken to protect components from moisture-related damage.

5. DETAILED REQUIREMENTS OR STATEMENTS

5.1 Printed board construction. This section specifies the general construction of the printed board when the engineering drawing does not otherwise indicate specific requirements. In cases where the engineering drawing deviates from the requirements listed herein, the engineering drawing shall prevail. In cases where the procuring document deviates from the requirements listed herein or the engineering drawing, the procuring document shall prevail. Only the applicable specifications should be referenced based on the design construction.

5.1.1 Rigid construction. Printed boards incorporating rigid dielectrics shall conform to IPC-6011 Performance Class 3 (high reliability) and IPC-6012 Class 3/A (military/aerospace applications) with the following exceptions to the IPC-6012 Class 3/A testing requirements:

Exception 1: PTH and Surface Solderability shall meet and be tested to Class 3 specifications.

Exception 2: Thermal Shock Testing: If the printed circuit board house is re-certified every 2 years, the testing shall not be required on a per design basis.

5.1.2 Flexible construction. Printed boards incorporating flexible dielectrics shall conform to IPC-6011 Performance Class 3 (high reliability) and IPC-6013 Type (1, 2, 3 ,5)* Use C (high temperature environment). * Type may vary by design.

5.1.3. Rigid/flex construction. Printed boards incorporating rigid and flexible dielectrics shall conform to IPC-6011 Performance Class 3 (high reliability), IPC-6012 Class 3/A (military/aerospace applications) Type 1, 2 or 3 and IPC-6013 Type 4* Use C (high temperature environment). * Type may vary by design with the following exceptions to the IPC-6012 Class 3/A testing requirements:

Exception 1: PTH and Surface Solderability shall meet and be tested to Class 3 specifications.

Exception 2: Thermal Shock Testing: If the printed circuit board house is re-certified every 2 years, the testing shall not be required on a per design basis.

5.1.4. Flex w/stiffener construction. Printed boards incorporating flexible dielectrics with stiffener added shall conform to IPC-6011 Performance Class 3 (high reliability), IPC-6012 Class 3/A (military/aerospace applications) Type 1, 2 or 3 and IPC-6013 Type 2* Use C (high temperature environment). * Type may vary by design with the following exceptions to the IPC-6012 Class 3/A testing requirements:

Exception 1: PTH and Surface Solderability shall meet and be tested to Class 3 specifications.

Exception 2: Thermal Shock Testing: If the printed circuit board house is re-certified every 2 years, the testing shall not be required on a per design basis.

5.1.5 Rigid dielectric. Rigid dielectric shall be per IPC-4101 L24XXXXH1/H1CA.

Where “XXXX” indicates the board thickness specified on the drawing.

The vendor shall choose the internal stack-up thicknesses to achieve the overall board thickness specified.

Where the H’s in “H1/H1” indicate Metal Cladding Type. “Copper, electrodeposited, HTE (high temperature elongation) per IPC-4562 grade 3”.

Where the 1's in "H1/H1" indicate the copper thickness – 1 indicates an area weight of 1 oz cu/sq ft. with an average thickness of .00135". Designs requiring a thickness other than 1 oz cu shall be specified on the master drawing.

5.1.6 Stiffener. Rigid dielectric used as a stiffener shall be per IPC-4101 L24XXXXO/O.

Where "XXXX" indicates the stiffener thickness specified on the drawing.

5.1.7 Flexible base dielectric (non-clad). Flexible non-clad dielectric shall be per IPC-4202/1 E1E1 (if used).

5.1.8 Adhesive-coated dielectric. Cover layer shall be per IPC-4203/1 E1E1M2/0. Adhesive thickness as required based on the copper thickness it is being applied over. Cover layers shall not extend more than 0.050" into rigid sections. Some existing designs may require the cover layers to extend further into the rigid sections due to plated holes or vias near the edge of the rigid section. If cover layer gerbers are not supplied the board house shall apply the required cover layers as described here.

5.1.9 Flexible metal-clad dielectric. Flexible metal-clad dielectric shall be per IPC-4204 /11 E1E2O0 CU-W7-1S/1S. Designs requiring a thickness other than 1 oz cu shall be specified on the master drawing.

5.1.10 Adhesive. Adhesive shall be per IPC-4203/18 O0O0M1 with thickness as required.

5.1.11 Adhesive prepreg. Adhesive shall be per IPC-4101/24 with thickness as required.

5.1.12 Plating. Plating on flex layers shall be limited to holes and pads only (spot plate).

5.1.13 Etchback. As required per design stackup, etchback shall be between 5 um [197 uin] and 40 um [1574 uin]. Negative etchback shall not be allowed.

5.1.14 Solder mask. Solder Mask Over Bare Copper (SMOBC) shall be in accordance with IPC-SM-840, Class H.

5.1.15 Final finish. All exposed conductors shall be solder coated using Hot Air Leveling (HAL) (or HASL) per IPC-6012C-2010 Type S per J-STD-003 and J-STD-006, composition SN60A or SN63A, Type S Finish - Solder Coating over Bare Copper. Thickness = Coverage & Solderable

5.1.16 Strain relief. A small bead of epoxy adhesive shall be applied at all rigid/flex transition zones on both sides of the flex as required per PD1635353.

5.1.17 Silkscreen. Mark component symbols (legend) per IPC-4781 Type 2 color, white.

5.1.18 Identification marking. Mark board with vendor logo, date code, and CLSS part number and engineering drawing revision level, on bottom side of board unless otherwise indicated on drawing. Marking shall conform to IPC-2221.

5.1.19 Dielectric strength. Dielectric strength shall be in accordance with IPC-TM-650 paragraph 2.5.7 condition B.

5.1.20 Hole centers. All holes (not dimensioned) shall be centered on pads and have a minimum annular ring per IPC-6012 Class 3.

5.1.21 Profile features. Profile features not dimensioned are derived from the electronic gerber files supplied and shall conform to the profile tolerance indicated on the drawing.

5.2 Carrier/array. CLSS engineering drawings reference individual printed board profiles. The contract manufacturer (board assembler) may require boards be produced in arrays and/or carriers to best meet their assembly process requirements. If required, the purchase order should detail the required information.

5.3 Breakaway features. Breakaway features are features that allow individual boards to be separated from one another in an array or from a carrier. These features shall meet the requirements detailed in 5.3.1 or 5.3.2.

5.3.1 Scoring. Scoring shall be the preferred method used to separate boards. Scored lines should have a remaining web thickness in the score of 0.012 inch to 0.016 inch to facilitate separation without damage to the assembly. The score lines shall be equally deep on both sides of the printed board, and the point of the score line grooves shall be in line with each other within 0.002 inch. See figure 1 for further illustration. Score angles over 45 degrees shall only be used with written approval of CLSS Engineering. Score lines shall be applied using a router or similar low-stress material removal means. Score lines shall not be applied using a material displacement method without written approval of CLSS Engineering.

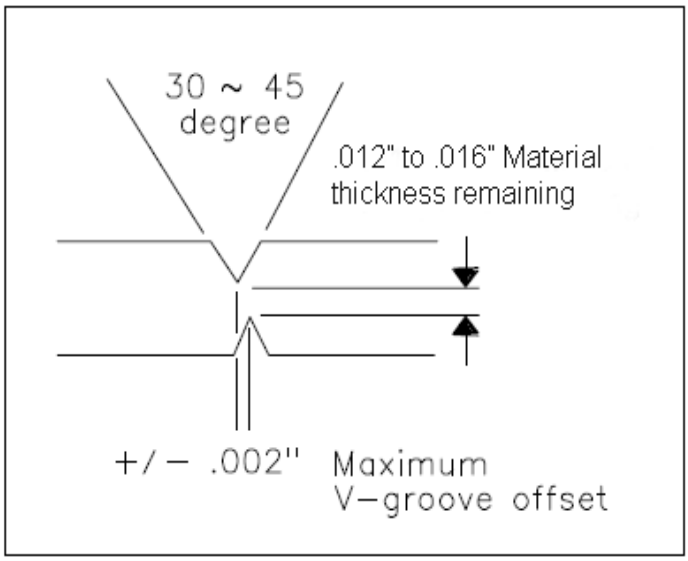

Figure 1. Scoring Detail

5.3.2 Breakaway tabs. The use of breakaway tabs (or mouse bites) shall only be used with written permission from CLSS Engineering. When breakaway tabs are allowed, the tabs shall leave a smooth edge after the break; in no instance shall jagged material protrude from the edge of the board. The tabs shall break or separate with minimal force to prevent damage to the board and assembled components as high stress concentrations exist at the breakaway site.

5.4 Packaging. To prevent moisture related issues during soldering, the circuit boards must be sufficiently dry prior to packaging. The Maximum Acceptable Moisture Content (MAMC) of packaged boards shall be 0.1% moisture weight to resin weight. The board fabricator is encouraged to develop a test methodology and statistical process control plan to ensure the MAMC of the packaged boards. The moisture content of a printed board may be determined using IPC-TM-650, Method 2.6.28. The preferred method to ensure MAMC prior to packaging is to control the uptake of moisture during the fabrication process. If bake out is required to achieve

MAMC, the guidelines of IPC-1601 should be used. Boards shall be packaged in an outer barrier bag with desiccant and humidity indicator card included.

5.4.1 Outer barrier bag. Boards shall be packaged in a material that meets J-STD-033 requirements. Prohibited bag materials include clear plastic and polymers that do not contain a metallic film, as these have been found to provide limited moisture barrier performance. Bags shall be sealed as to leave enough excess material to allow for a minimum of one resealing of the moisture barrier bag to facilitate incoming inspection. Examples of acceptable bags include.

A. Nylon/ Foil/ Polyethylene. Outer layer of Electrostatic Discharge (ESD) nylon, aluminum foil middle layer and an inner layer of polyethylene. Example material is 3M Dri-Shield Moisture Barrier 3400.

B. Tyvek / Foil / Polyethylene. Outer layer of Tyvek with aluminum foil middle layer and an inner layer of polyethylene.

C. Aluminized Polyester / polyethylene. Acceptable material when used with the material thickness greater than 0.18 mm (0.007 in).

5.4.2 Desiccant. Desiccant material quantity and quality used in the internal packaging shall be in accordance with IPC/JEDEC J-STD-033 and shall be sulfur free. The desiccant materials shall be placed on the edges of the Printed Circuit Board (PCB) so as not to potentially cause board warping if packages are stacked within the shipping carton. This placement also will prevent the desiccant from interaction with the soldering finish surfaces.

5.4.3 Humidity Indicator Card (HIC). An HIC shall be packed inside the outer moisture barrier bag along with the desiccant. The HIC shall be in accordance with IPC/JEDEC J-STD-033 and be sulfur free.

5.4.4 Laminate witness coupon. If requested by the board assembler, the board fabricator shall supply a witness coupon in the packaging that can be used to verify MAMC of the packaged boards per IPC-1601. The witness coupon shall be of same construction and materials and undergo the same moisture bake out and packing procedures as the boards.

5.4.5 Moisture caution label. Board packaging shall be marked with a moisture sensitivity caution symbol per IPC-1601.

5.4.6 Moisture exposure label. A moisture exposure label shall be affixed to the outer packaging. The moisture exposure label provides a form for logging the exposure time of the printed boards to environments of greater than 5% Relative Humidity (RH). The label shall provide fields for logging the time the packaging's moisture seal is opened, is closed, and the calculated total time of exposure. The board fabricator is exempt from supplying this label; however, the label shall be applied as soon as practical on incoming inspection/receiving.

5.5 Board inspection. The CLSS (or designated) inspector shall verify at a minimum the following information prior to accepting the received boards.

A. Verify the IPC qualification and performance specification meets the specified requirements.

B. Verify that the board materials meet the specified material callouts.

C. Verify that at a minimum the CLSS part number, drawing revision, vendor logo/name and manufacturing date code are marked on each individual board.

D. Verify that the board packaging is received in an approved moisture barrier bag with a moisture sensitivity caution symbol, desiccant, and HIC.

E. Verify that the HIC does not indicate 10% or greater RH.

5.5.1 Moisture. To minimize printed board exposure to moisture, the moisture barrier packaging should be open for no longer than 15 minutes to facilitate incoming inspection. The desiccant shall be replaced with a fresh desiccant and the packaging shall be resealed. If the HIC indicates 10% or greater RH then the bag shall be resealed with a fresh desiccant and fresh HIC and the outer packaging marked “BAKE OUT REQUIRED PRIOR TO SOLDERING”. If the HIC indicates 60% or greater RH or a HIC is not present in the packaging then the boards shall not be used for saleable product as it cannot be assured the level of moisture exposure.

5.6 Storage and handling. Boards shall be stored in their original packaging, re-packed per paragraph 5.4, or stored in a dry cabinet with a maximum 5% RH environment. Regardless of storage means, the moisture exposure label shall remain with the boards at all times. Handling requiring boards to be exposed to 5% RH environments or greater shall be marked on the moisture exposure label.

6. NOTES

Changes to this document shall be made using the applicable practices described in ST1637815. Changes shall be reviewed and approved in accordance with Section 9 of ST1637815.