2.0 OBJECTIVE & SCOPE:

2.1 Objective: To communicate DCA Manufacturing's requirements and expectations for Rigid Printed Circuit Boards to Rigid Printed Circuit Board Suppliers.

2.2 Scope: Covers all rigid printed circuit board suppliers.

2.3 Effective date: DCA purchase orders issued February 18, 2013 and thereafter, refer to paragraph 5.4.7 for exceptions for existing tooling.

3.0 RESPONSIBILITIES:

3.1 Printed Circuit Board Supplier: Contact respective DCA Account Manager for all written authorizations, deviations, clarifications, and approvals prior to production.

3.2 DCA Manufacturings (DCA) Engineering: Identify rigid printed circuit board requirements for DCA manufacturing.

3.3 DCA Account Managers:

3.3.1 Ensure PCB suppliers have current work instruction requirements.

3.3.2 The point of contact for PCB Suppliers.

3.3.3 Obtain all necessary internal approvals prior to responding in writing to a PCB Supplier request.

3.4 DCA Quality: Audit to the requirements of this work instruction.

3.5 DCA Receiving Inspection: Receive and inspect to this work instruction and record data as required.

3.6 DCA Manufacturing Units: Report and assist with problems or issues for PCB Suppliers or Customer product with respect to this work instruction compliance.

3.7 Standards: PCB Industry recognized standards such as: IPC, JDEC J-STD, ANSI Standards.

4.0 DEFINITIONS:

4.1. C of C: Certificate of Compliance.

4.2. Date/Lot Code: Represents the date the production of the PCB lot is started. A date code is a unique number identifying when each production run is started.

4.3. Documentation Package: DCA Specifications (this document), DCA Array/Rail Drawing, Customer Gerber File, Customer Artwork with Fabrication Notes, Customer PCB Specifications as noted in Customer Artwork.

4.4. ENIG: Electroless nickel immersion gold

4.5. HASL: Hot Air Solder Level

4.6. Production Panel: The PCB Supplier's processing panel.

4.7. Restart: Production started to make up for production fall-out, adds, or other build interruptions that causes PCB losses that need to be replaced.

4.8. DCA: DCA Manufacturing.

4.9. Node: A connection point between a network and an off-PWB item. The off-PWB item may be permanently attached (electrical component packages) or a temporary contact (test probe). Nodes are electrical contact points such as plated holes, SMD lands, or loaded board test points.

4.10. General inspection report: A report covering the specified and actual requirements of material used, layers, soldermask type, silk screen type, etc. Often referred to as a first article inspection report.

4.11. Cross-section report: A report that documents the thicknesses of each layer, CU thickness, inner-connect defects, etc. Often referred to as a microsection report.

5.0 PROCESS:

5.1 Documentation Control

5.1.1 Order of Documentation precedence

5.1.1.1 DCA Specifications.
*NOTE: At any time, customer specifications may override DCA Specifications.

5.1.1.2 DCA Drawings

5.1.1.3 DCA Purchase Order.

5.1.1.4 Artwork.

5.1.1.5 Customer Engineering Drawings.

5.1.1.6 IPC-A-600 and IPC 6011/6012.

5.1.2 Upon documentation review, suppliers shall notify DCA in writing of any of any Standards that are obsolete, review discrepancies and list the current Standards.

5.1.3 For compliance questions regarding documentation issues of the following, request clarification in writing.

5.1.3.1 Not able to meet DCA or Customer specifications.

5.1.3.2 Contradictions (DCA and Customer specifications are different), Order of Precedence.

5.1.3.3 Missing information identified in definitions or documentation package.

5.1.4 No changes in materials will be made without prior written notification.

5.1.5 No outsourced processes will be made without prior written notification.

5.1.6 No documentation changes (Gerber, CAD, Drill files, etc.) will be made without prior written approval.

5.1.7 No Changes in the place of manufacture will be made without prior written approval.

5.1.8 Individual DCA manufacturing facilities may add special requirements, which will be referenced on the Purchase Order and/or DCA Drawings.

5.2 Arrays

5.2.1 The DCA drawing must be followed if provided. If a DCA drawing is not provided, or available, the listed information 5.2.6 through 5.2.15 is required.

5.2.3 Breakaways can have supplier processing panel robbers on them, no burrs or high spots that are above the board surface are allowed.

5.2.4 Quote the PCB per documentation provided. Supplier may also submit an alternative quote with suggestions for lower PCB cost.

5.2.5 If Copper thieving is used it must not be the same size and shape as the Fiducials.

5.2.6 PCB Tooling Holes on the breakaway are not required by DCA but may be added at the PCB Supplier's discretion.

5.2.7 Fiducials (filled circle) are required on both sides of an array, three fiducials per array. Fiducials must be 1.0mm (0.039 inch) diameter with a suitable mask relief. The center of the Fiducial should be located 0.400 x 0.200 inches from the corners of the panel. The preferred corners for the top side are lower left, upper right, and upper left.

5.2.8 Fiducial spacing for the top side and bottom side should be the same but have mirrored symmetry (ie, lower right, upper left, and upper right)

5.2.9 The minimum breakaway width is 0.300 inches for panels with scored separation and 0.350 inches for panels that include routed separation. Rails can be made larger to provide additional rigidity as long as it does not adversely affect the yield from the production panel.

5.2.10 The minimum Y dimension for a panel is 4.000 inches. The X dimension should be at least 75% of the Y dimension, and is preferred to be greater than the Y dimension.

5.2.11 When sizing a multi-board panel the Y dimension should be kept as small as possible while maintaining the 4.000 inch minimum. The panel array should be optimized to provide the best yield from the production panel.

5.2.12 A 1.000 x 0.250 silkscreen box free of thieving should be added to the strip on the top side 2.000 x 0.050 from the lower left corner.

5.2.13 A 1.000 x 0.250 area free of thieving should be provided on the strip on the bottom side 2.000 x 0.050 from the lower right corner.

5.2.14 The preferred method of panel separation is scoring. The specification of the V-score can be seen in Figure 5.2.14 shown below. The orientation of the boards in the panel should provide a minimum number of horizontal scores.

Scoring Specifications			
Board Thickness	Score Web Thickness	Tolerances	Displacement
0.031"	0.010	+/- 0.003	+/- 0.005
0.062"	0.012	+/- 0.003	+/- 0.005
0.093"	0.022	+/- 0.005	+/- 0.005
0.125"	0.030	+/- 0.005	+/- 0.005

5.2.15 If a board will have components that are within 0.040 inches of the edge or will hang over the edge, the corresponding edges should be routed instead of scored. It is preferred to have the route slots oriented vertically on the panel with no tabs.

5.3 PCB Dimensional Requirements

5.3.1 Maximum Bow and Twist shall be 0.75% or less for boards that use surface mount components and 1.5% or less for all other boards. Determined by physical measurement and percentages in accordance with IPC-TM-650, Method 2.4.22. Panels that contain multiple printed boards, which are assembled on the panel and later separated, shall be assessed for Bow and Twist in panel form.

5.3.2 All un-plated 3.18mm (0.125 inch) hole diameters will have a manufacturing tolerance of –0.000mm, +0.076mm (-0.000, +0.003 inch), unless the customer manufacturing tolerance is less.

5.4 Traceability

5.4.1 The Date Code of each individual board will be in copper unless specifically requested to be in silk screen on the end customer documentation.

5.4.2 The Date Code represents the date that production is started. It must include the year and the week of manufacture.

5.4.5 Manufacturer's identification logo with facility designator is required in copper or silkscreen on each PCB and must be legible.

5.4.6 No boards, with the copper Date Code can be over six months old or supplied unless prior written approval (internally DCA will secure approval from all of the following: DCA Account manager, Process Engineering PCB representative, and Quality PCB representative).

5.4.7 Existing tooling purchased under the previous work instruction WI 4.4-745-001 revision 07/31/02 will have a deviation to Items 5.4.2 through 5.4.4. Upon release of ECO or tooling updates, the requirements of this document and revision must be implemented.

5.5 Exposed Surface Features

5.5.2 Unless otherwise specified, the Lateral Conductor Spacing is 100 micro-meters (0.004 inch) minimum per IPC-6012, Table 1.2.

5.6 Solder Mask

5.6.1 Unless otherwise specified, the solder mask will be green Liquid Photo Imageable and finish will be matte, Solder mask must conform to IPC-SM-840 Class T for assemblies that are IPC performance class 2, and IPC-SM-840 Class H for assemblies with that are IPC performance class 3. Preference is for a matte finish when several mask options are specified.

5.6.2 No solder mask can be on any exposed copper areas as defined within the Gerber or Cad.

5.6.3 If via masking under any array package (BGA) is required; a minimum of 100% coverage of the annular ring by the masking is required.

5.6.4 The arrayed PCBs must be supplied with solder mask dams between all pads, and annular rings. If these features are missing from the 1-up Gerbers they must be reconstructed in the array Gerbers.

5.7 PCB Finish

5.7.1 If a HASL finish is applied, the solder coating thickness must be 5.1 to 25.4 micrometers (200 to 1000 micro-inches) over 90% of the SMT land, see Figure 5.7.1 below.

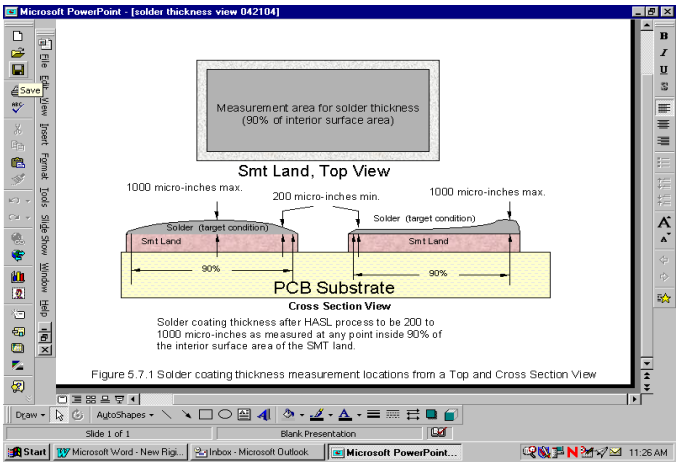

Figure 5.7.1 Solder coating thickness measurement locations from a Top and Cross Section View

5.7.2 A HASL solder finish containing lead will be a Tin/Lead (Sn63/Pb37) type C alloy as specified in J-Std-006, number 3.3., unless otherwise specified by customer documents.

5.7.3 If no-lead finish is required but the type is not specified, the preferred type is ENIG (Electroless Nickel Immersion Gold).

5.8 Nomenclature Ink

5.8.1 Nomenclature ink (silkscreen) must be non-conductive and permanent, unless otherwise specified.

5.8.2 If nomenclature covers any exposed electrical network conductive features, contact DCA in writing for directions. Do not proceed until you obtain written direction.

5.8.3 Nomenclature ink thickness must be less than or equal to 0.025 mm (0.001inch).

5.8.4 Nomenclature markings must meet IPC-A-600, section 2.8 Marking.

5.9 Solderability

5.9.1 The Supplier must warranty the final finish plating (or surface protectant) applied meets the manufacturer's requirements for solderability and has a six month shelf life from the date of application.

5.10 Cleanliness

5.10.1 Boards must be visually free of particulate material (per IPC -A-600, section 5.0) such as:

5.10.1.1 Flux Residues.

5.10.1.2 Particulate matter.

5.10.1.3 Chemical salt residues

5.10.1.3 Finger prints.

5.10.1.4 Corrosion (oxides).

5.10.1.5 White residues.

5.10.2 Boards must have less than 0.8 micrograms/square cm (5.0 micrograms/square inch) Sodium Chloride (NaCl) equivalent ionic contamination or ionizable flux residue. Cleanliness requirements above shall be per IPC 6012B, section 3.9 and 3.9.2.

5.11 Electrical Testing

5.11.1 Test requirements see table 5.11.1, reference IPC-9252, 4.3 Test parameter Matrix.

Table 5.11.1 Test Requirements

Line	Level	A	B	C
1	Class	1	2	3
2	Midpoint nodes	No	No	Yes
3	Surface Planes	One cut-out	One cut-out	All cut-outs
4	Non-network items*	No	No	Only if required by customer
5	Netlist Source	Golden PWB	CAM	CAD
6	Isolation	500kΩ	2MΩ	10MΩ
7	Continuity	50-1000Ω	50Ω	50Ω
8	Loaded board test points	No	Yes (if supplied)	Yes (if supplied)
8	All endpoint nodes	Yes	Yes	Yes
9	All Networks	Yes	Yes	Yes
*	* (i.e., alignment and mount holes, heat sink without solder mask, shielding on perimeter, Fiducials, thieving, other metal not in the design)			

5.11.2 100% Continuity and Isolation Electrical Testing required of all electrical connections (nodes), reference IPC-9252, Guidelines and Requirements for Electrical Testing of Unpopulated Printed Boards. Nodes tested must match a proven functional reference source, including but not limited to CAM/CAD digital data, master pattern artwork, or released drawings of boards.

5.11.3 An Electrical Test Stamp, applied during the electrical testing process as an indication of passing must be visible on each board.

5.11.3.1 Test stamps must not be placed on breakaways unless approved in writing by the DCA Account Manager.

5.11.3.2 If there is not sufficient space available on the PCB for a test stamp, a deviation must be obtained in writing from the DCA Account Manager.

5.11.3.3 Alternative marking methods possible if approved by DCA Corporate Quality.

5.12 PCB Shipping Packaging, General

5.12.1 Board Orientation must be the same in all PCB package bundles.

5.12.2 The Supplier must follow First In, First Out (FIFO) practices for Date/Lot Code shipments. Supplier's shipping product with Date/Lot Codes prior to the last Date/Lot Code received will be subject to lot rejection, unless prior written approval is granted by the Account Manager.

5.12.3 ESD approved materials must be used for wrapping PCBs and leave no residue on the PCBs. Some "Pink Poly Bags" leave a contaminating residue on the PCBs and must not be used.

5.12.4 All boards must follow all guidelines set forth in the current revision of IPC-1601.

5.12.4.1 All boards must be delivered in dry packaging. Dry packaging is defined by IPC-1601 as "packaging that consists of desiccant material and a Humidity Indicator Card (HIC) sealed with the printed boards inside a Moisture Barrier Bag (MBB)."

5.12.4.2 An example of a Humidity Indictor Card can be found in the specification IPC/ JDEC J-STD-033A, number 3.3.2.3 Humidity Indicator Card, see Figures 3-1 and 3-2.

5.12.4.3 Desiccant containers should be placed along one side of each PCB stack before sealing.

5.12.4.4 When PCB stack arrangements are sealed in multiple stacks, sealing should allow at least 12.7 mm (0.5 inch) of clearance between each stack .The edges of the vacuum tray must maintain an airtight seal and create a cushioning bubble border.

5.12.4.5 PCBs with holes greater than 6.4 mm (0.25 inch) must have the top and bottom PCB in the stack covered with slip paper that is sulfur free.

5.12.4.6 PCB packages received with tears, rips, or pinholes, are subject to rejection and return.

5.12.4.7 Shipping box may contain multiple packages of a single part number however each package shall contain only one Date Code. Quantity and Date Code must be marked on the package and the box.

5.12.4.8 Shipping boxes must be marked with the total number of packages, quantities, and Date Code of packages contained within.

5.12.4.9 The maximum weight of a box must be equal to or less than 25 pounds. Over pack container for boxes is allowed, however individual box weight must not exceed stated maximum.

5.12.4.10 No Styrofoam, peanuts, chips, or any small particle or static generating material can be used for dunnage or padding in shipping boxes.

5.13 Miscellaneous

5.13.1 When an Edge Card Connector is on the edge of a PCB and is in line with a breakaway outside edge, the Chamfer of the Edge Card Connector may extend into the breakaway a maximum of 6.4mm (0.25 inches), the balance of the breakaway edge must be square.

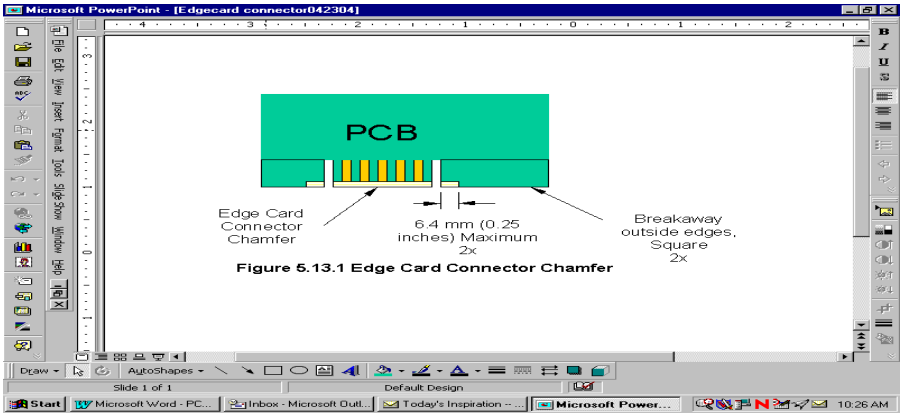

Figure 5.13.1 Edge Card Connector Chamfer

5.14 Repair and Rework

5.14.1 Repair: Welds are not allowed for repairing etch-outs or opens.

5.15 Certificate of Compliance

5.15.1 A Certificate of Compliance (C of C) shall be available upon request or as specified by the customer's documentation.

5.16 First article requirements

5.16.1 First Articles will be addressed on the purchase order as required.

6.0 QUALITY DOCUMENTS/RECORDS:

Refer to Master List of Quality Records for Retention Time & Location

6.1. All lot shipments must come with the following:
(1) General inspection report
(2) Cross section report for boards with four or more layers.

If the cross-section report has a high quality, blown-up view of the cross section, a coupon does not need to be provided.

6.2 Exception may be given by the DCA quality manager.