Axiom PCBA Design for Manufacturability Guidelines

Section: 1.0	Revision: B	Revision Date: 10/31/18

DFM Subject: Purpose, Scope and Table of Contents

1.1 Purpose

This document is used to communicate manufacturing capability and design for manufacturing guidelines for printed circuit board assembles (PCBA) that contain surface mounted and through-hole components. Updates will be issued on a continuous basis to address changes in technology and manufacturing capability.

1.2 Scope

This document covers manufacturing capability and design for manufacturing for printed circuit board assembly. Industry standards are used wherever possible. Where industry standards are not available best industry practice is used.

1.3 Design for Manufacturing

Designing for manufacturability (DFM) is the art and science of designing a printed circuit board assembly for ease of manufacture. DFM will decrease product cost, reduce assembly defects and improve reliability.

1.4 Table of Contents

1.0 Purpose, Scope and Table of Contents
2.0 Terms and Definitions
3.0 Industry Standards and Axiom Documents
- 3.1 Industry Standards That Support DFM

4.0 PCBA Assembly Overview
- 4.1 Manufacturing Flow
- 4.2 Solder Paste Printing
- 4.3 Component Placement
- 4.4 Reflow soldering
- 4.5 Cleaning (flux removal)
- 4.6 Through Hole Insertion
- 4.7 Wave and Selective Soldering
- 4.8 Mechanical Assembly
- 4.9 Inspection and Test

5.0 Component Guidance, Surface Mounting
- 5.1 Component Packaging
- 5.2 Moisture Sensitive Devices
- 5.3 Capacitors
- 5.4 Resistors
- 5.5 Connectors
- 5.6 Sockets
- 5.7 Fuses

- 5.8 Integrated Circuit (IC) Packages
- 5.9 Transistors and Diodes
- 5.10 Resistor Networks (R-packs)
- 5.11 Surface Finish Options

6.0 Component Guidance, Through-Hole Mounting
- 6.1 Through-Hole Component Usage
- 6.2 Lead Protrusion
- 6.3 Resistors and Capacitors
- 6.4 Integrated Circuit (IC) Packages
- 6.5 Sockets
- 6.6 Two Way Switches
- 6.7 Delay Buttons
- 6.8 Batteries
- 6.9 LED Devices
- 6.10 Transistors and Diodes
- 6.11 Connectors
- 6.12 Surface Finish Options

7.0 Component Packages and Naming Conventions
- 7.1 Approved Surface Mount Components
- 7.2 Approved Through-Hole Mount Components
- 7.3 Component Naming Conventions
- 7.4 Reference Designator Marking Requirements
- 7.5 Formula for Secondary Side Mounting

8.0 Printed Circuit Board (PCB) Guidance
- 8.1 PCB Fabrication and Surface Finish
- 8.2 PCB's per Panel and PCB Size

9.0 Common PCB DFM Issues
- 9.1 Material Type
- 9.2 Material Stackup
- 9.3 Trace Width and Space
- 9.4 Via Design (through-hole, blind and buried)
- 9.5 Impedance
- 9.6 Acid Traps
- 9.7 Duplicate Drill Holes
- 9.8 Insufficient Annular Ring
- 9.9 Missing Solder Mask
- 9.10 Missing Paste Mask
- 9.11 Edge Violation
- 9.12 Solder Mask Slivers
- 9.13 Copper Slivers & Islands
- 9.14 Silkscreen-to-Pad Violations
- 9.15 Copper Pad-to-Cutout
- 9.16 Component-to-Outline

10.0 DFM Requirements, PCB and PCBA
- 10.1 PCB Size and Thickness (Max and Min)
  - 10.1.1 PCB Size and Thickness, Pad Cratering Prevention

10.1.2 BGA Size and Location, Pad Cratering Prevention
10.2 PCB Edge Clearance and Direction of Travel
10.3 PCB Tooling Hole Size and Location
10.4 Mounting Holes Used For Grounding
10.5 Fiducial Size and Location
10.6 Selective Soldering Clearance
10.7 Thermal Relief Connections
10.8 PCB Multi-Pack Framing and Removable Rails
10.9 PCB V-Scoring and Drilled Breakaway Tabs
10.10 Soldermask Defined Lands
10.11 Soldermask Clearance for Retention Clips
10.12 Gold Plated Edge Connectors and Grounding Strips
10.13 Drilled Via Pad Stack and Hole Size Recommendations
10.14 Lead to Hole Ratio Recommendations
10.15 PCB Balanced Layer Construction
10.16 Via Placement and Spacing
10.17 Via Soldermask Encroachment
10.18 Via Plugging and Filling With Soldermask
10.19 Copper Feature Edge Clearance
10.20 SMT Component Height Restrictions
10.21 Component Assembly Spacing
10.22 BGA Inspection and Rework Clearance
10.23 Through-Hole Connector Orientation
10.24 Press Fit Connector Requirements
10.25 Heatsink Attachment, Pad Cratering Prevention
10.26 High Density Interconnect Pad Stack Geometries
11.0 DFM Requirements, Land Pattern Design
11.1 Land Pattern Design, IPC-7351
11.2 Land Pattern Design, Resistors and Capacitors
11.3 Land Pattern Design, Gull Wing Leads
11.4 Land Pattern Design, J Leads
11.5 Land Pattern Design, BGA
11.6 Land Pattern Design; Leadless and Thermal Lands
12.0 DFT Requirements, Probe Testing
12.1 Test Coverage
13.0 DFX Scorecard
13.1 DFX Scorecard Template