1	Tracking Information				
PCB Part Number: D3005H100-D01			**Issue Level:** A		
PCB Description: M9 Portable Tester			**Project Title:** M9		
2	**Specification Requirements** – *choose only one from each category.*				
GENERAL QUALITY STANDARD	☒ IPC 600E CLASS 2	☐ OTHER:			
BOARD MATERIAL	☒ FR-4	☐ OTHER:			
BOARD THICKNESS, +/- 0.007”	☒ 0.062”	☐ 0.093”			
SIGNAL LAYERS	☐ 2	☒ 4	☐ 6	☐ 8	☐ 10
FINISHED COPPER THICKNESS	☒ 2 OZ. NOMINAL	☐ 3 OZ. NOMINAL	☐ OTHER:		
PCB ARRAY	☒ INDIVIDUAL BOARD	☐ OTHER:			
SURFACE FINISH	☒ HOT AIR SOLDER LEVEL (HASL)	☐ IMMERSION GOLD	☐ IMMERSION SILVER		
SOLDER MASK TYPE	☒ LPI-GLOSS	☐ LPI-MATT			
SOLDER MASK COLOR	☒ GREEN	☐ BLACK	☐ RED	☐ OTHER:	
SOLDER MASK SIDES	☒ COMPONENT SIDE & SOLDER SIDE	☐ OTHER:			
SILK-SCREEN LEGEND COLOR	☒ WHITE	☐ OTHER:			
SILK-SCREEN LEGEND SIDES	☐ COMPONENT SIDE	☐ SOLDER SIDE	☒ COMPONENT SIDE & SOLDER SIDE		
DATE CODE AND LOGO	☒ YES	☐ NO			
3	**Approval**				
Submitted By (First Last): Steve Guo		**Approved By (First Last):** Steve Guo		**Date (MM/DD/YYYY):** 7/9/2019	
4	**Other**				
Location of PADS/Gerber Files: \\usepblfil01.railad.com\Drafting\MECH\PADS\PCB\					

Instructions

To successfully complete the PCB Specification Fabrication Process, follow the instructions below:

1. The Design Engineer electronically fills out the PCB Fabrication Specification Form and includes it as part of the initial ECO submittal.
2. The Design Engineer and Drafting may rework the file during the design process. The final version of the form will be stored with the closed ECO.
3. The Design Engineer renames the final version of the electronic form to include the PCB part number and copper issue letter (e.g., D7710H0123-D01_A.docx) and then generates a PDF with the same filename.
4. Drafting stores the PDF in the internal storage system and includes it with the PADS/Gerber files that are then sent to the vendor for purchasing.

Definitions

The table below provides an explanation of the terms found throughout the specification form:

PCB part number: This is the part number of the PCB. It should be followed with -D01. “D” stands for detail.	**Finished Copper Thickness:** We normally use 2 OZ. copper. PCB designers use heavy copper when designing and manufacturing PCBs to increase current-carrying capacity as well as resistance to thermal strains.
PCB description: The name of the PCB (e.g., Seattle ADU Micro PCB).	**Surface Finish:** We typically use hot air solder level (HASL). ENIG stands for (Electroless nickel immersion gold). HASL is less environment-friendly than ENIG. HASL is cheaper than ENIG. HASL has an excellent solder-ability and good shelf life while ENIG has good corrosion resistance. ENIG is good for aluminum wire bonding, excellent for fine-pitch technology, has excellent solder-ability and excellent shelf life.
IPC 600E: (IPC) – Association Connecting Electronics Industries – Trade association aiming to standardize the assembly and production requirements of electronic equipment and assemblies.	**Solder Mask Type:** We typically use LPI-gloss. LPI-gloss vs LPI matte. Gloss solder mask is light reflective and lighter in color. Matte will have no shine and appear darker.
Class 2: A Class 2 printed board is one in which continued performance, extended life, and uninterrupted service is desired but not absolutely critical. (e.g., Motherboard in a home PC would typically be Class 2).	**Solder Mask Color:** We typically use Black and Green. This is basically the color of the PCB.
FR-4: FR4 is a grade designation assigned to glass-reinforced epoxy laminate PCBs. FR-4 is a composite material composed of woven fiberglass cloth with an epoxy resin binder that is flame resistant (self-extinguishing).	**Approval:** This is an electronic signature.
Board Thickness: We use either 0.062” or 0.093” depending on the application. Consult with the project engineer. Normally, a thicker board (0.093”) is used to maintain mechanical structure.	**Location of PADS/Gerber Files:** Location of the ZIP folder containing the PADS/Gerber files on the network drive.