

1. Compliance to standards

- Printed circuit board shall be manufactured as per IPC-6012 Class 2.
- Printed circuit board shall be RoHS compliant.
- Printed circuit board shall be UL796 compliant.
- Printed circuit board shall meet UL94 V-0 flammability rating.
- The acceptability of the printed circuit boards shall be evaluated according to IPC-A-600 Class 2.

2. Board dimensions / tolerances

- Board dimensions shall be as specified in the drill drawing Gerber file.
- Board dimensions are in mms. Tolerances are .XXX +/- 0.127mm

3. Silkscreen

- Silkscreen shall be non-conductive epoxy ink.
- Silkscreen shall be applied on top of soldermask.
- Silkscreen clearance with solderable copper patterns shall be of at least 0.005" (5 mil).
- Silkscreen shall be applied using a process that ensures that all printed characters are legible.
- Silkscreen clipping is permitted to achieve design and process requirements
- Silkscreen shall be applied using common thermal curing process to print the slikscreen text.
- Silkscreen shall withstand reflow and normal cleaning process.

PCB Requierements

REV DOC: A

PCB: PCB_F2838_Node.PcbDoc

DESIGNED: Alexis Maslyczyk DATE: 11/14/2022

4. PCB Stackup

Layer Stack Legend

Material	Layer	Thickness	Dielectric Material	Type	Gerber
	Top Overlay			Legend	GTO
	Surface Material				
	Top Solder	0.50mil	Solder Resist	Solder Mask	GTS
CF-004	Top Layer	0.71mil		Signal	GTL
Prepreg		6.18mil	PP-008	Dielectric	
CF-004	L2 - PWR	1.38mil		Signal	G1
Prepreg		9.84mil	PP-006	Dielectric	
CF-004	L3 - GND	1.38mil		Signal	G2
Prepreg		7.10mil	PP-021	Dielectric	
Copper	L4 - SIG HOR	1.38mil		Signal	G3
		9.84mil	FR-4	Dielectric	
Copper	L5 - SIG VER	1.38mil		Signal	G4
Prepreg		7.10mil	PP-021	Dielectric	
CF-004	L6 - AGND	1.38mil		Signal	G5
Prepreg		9.84mil	PP-006	Dielectric	
CF-004	L7 - PWR 3V3A 1V2	1.38mil		Signal	G6
Prepreg		6.18mil	PP-008	Dielectric	
CF-004	Bottom Layer	0.71mil		Signal	GBL
Surface Material	Bottom Solder	0.50mil	Solder Resist	Solder Mask	GBS
	Bottom Overlay			Legend	GBO

Total thickness: 66.78mil

5. Approval of manufacturing Gerber files and PCB

Manufacturing Gerber files and detailed PCB Stackup shall be sent to IHMC designer for approval before initiating the board fabrication process.