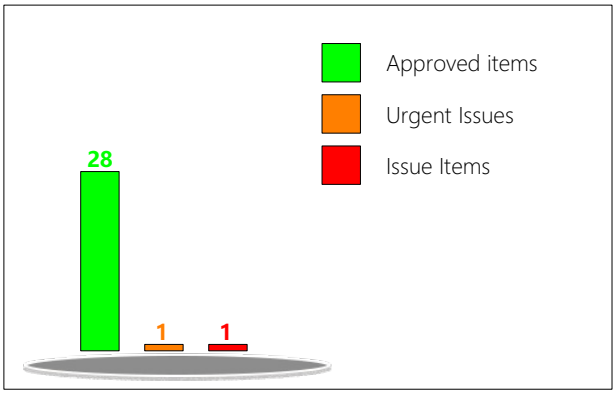
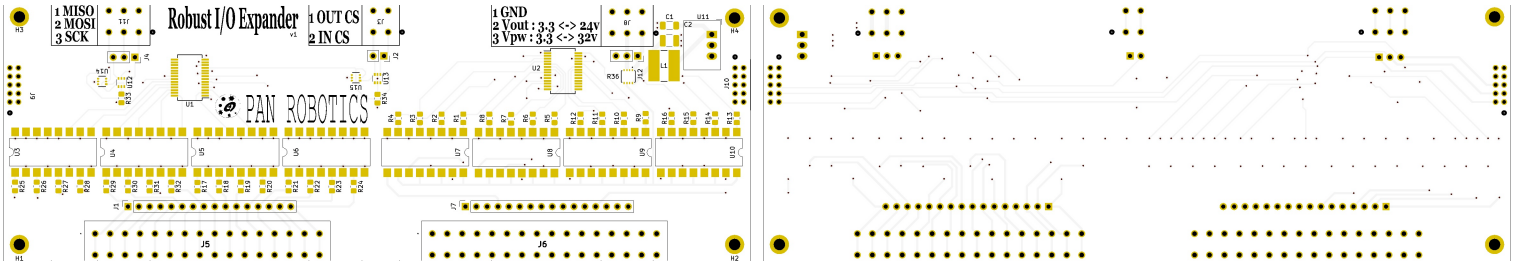
HQDFM Design for Manufacture(DFM) Report

File name: 2023-12-12

Time: 2023-12-12 Layer count:4 pcb thickness:1.60 Quantity5 mm

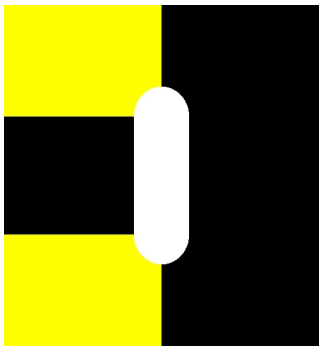
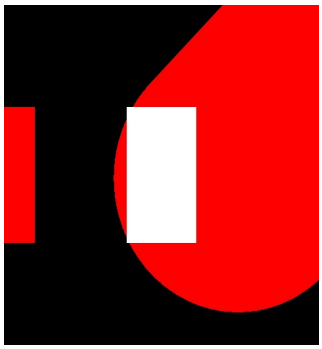


Basic Parameter Analysis	Trace Width/Spacing	10.00/8.00mil+
	Milling Density	45.5315m/m²
	Surface Finish Area	14.42%
	Test Point Count	437
	Panel Efficiency	85.1349%



Type	checklist item	checklist subitem	Result
PCB Trace Analysis	Open/Shorts (IPC)	1	Fail
	Signal Integrity	4	Pass
	Smallest Trace Width	1	Pass
	Smallest Trace Spacing	3	Pass 178
	Pad Spacing	2	Pass 16 ,Fail 52
	Min pad size	3	Pass 88
	Hatched Copper Pour	2	Pass
	RingHole	2	Pass 280
	Drill to Copper	5	Pass 272
	Board Edge Clearance	2	Pass 160
	Pad Hole	4	Pass
PCB Drilling Analysis	Hole Diameter	8	Pass 21
	Drill Hole Spacing	4	Pass 3
	Drill to Board Edge	4	Pass 37
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
	Drill Hole Errors	3	Pass
PCB Solder Mask Analysis	Solder Mask Spacing	2	Pass 64
	Missing SMask Openings	1	Pass
	Solder Paste Area	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Pass
ASS_component	Component Spacing	1	Fail
	Component to Edge	3	Fail
	Silk Spacing	0	Fail
	Pin mismatch	2	Fail
	Reference designator byte	0	Fail
	Plug-in Components	1	Pass

ASS_Pin Analysis	Pin-to-SMD Pad	5	Fail
	Through-hole pins	6	Fail
	Press-fit pins	4	Fail
ASS_Pad Analysis	Chip Pad	60	Fail
	Solder pad traces	4	Pass 416 ,Fail 5
ASS_Fiducials Analysis	Fiducials	1	Fail
	Fiducial Mark Analysis	3	Pass

ID	checklist item	Limits:	Value	Issue	image	Coordinate	Quantity	Level
1	Pad Spacing_Pad Spacing	6,8,10	0.20 mm	Pad to pad spacing for different SMD parts of 7.87mil were detected in your design. This could result in incomplete solder mask dams between the pads, thereby increasing the risk of solder bridges during assembly, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended to increase the spacing to at least 8 mil for green solder mask.		93.80,-94.04	52	Warning
2	Solder pad traces_pad_thermolysis_smt	-,-,-	Not Analyzed	If the wire occupies ≥85.85% of the solder pad's perimeter width ratio, rapid heat dissipation and a low solder paste melting point can lead to cold solder joints. It is recommended to optimize the trace routing and the ratio of copper connected to the solder pad.		200.11,-92.68	1	Risk