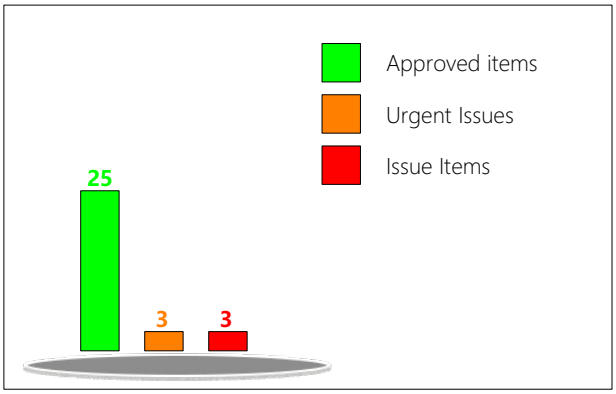
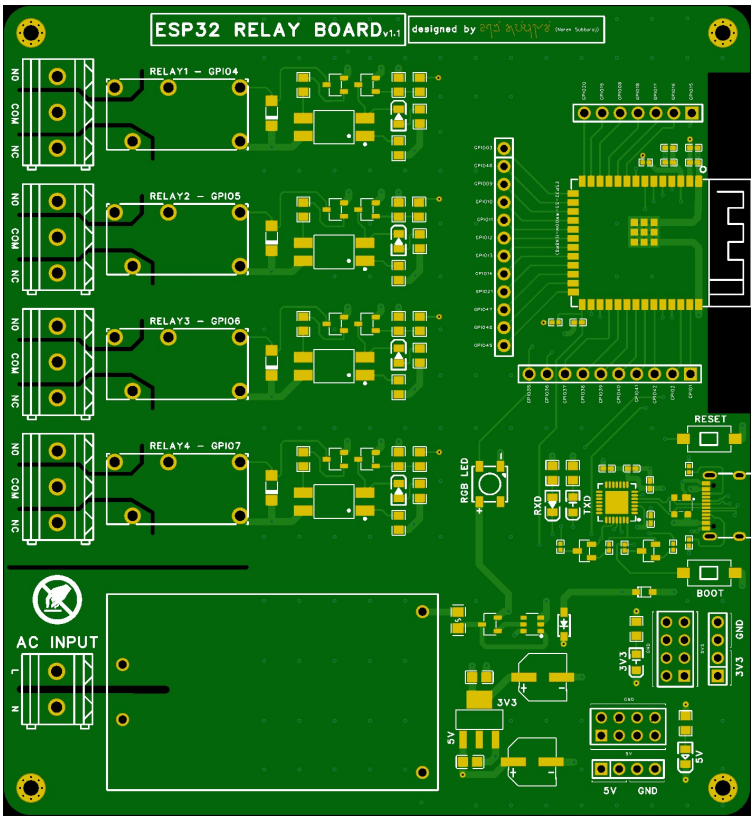
HQDFM Design for Manufacture(DFM) Report

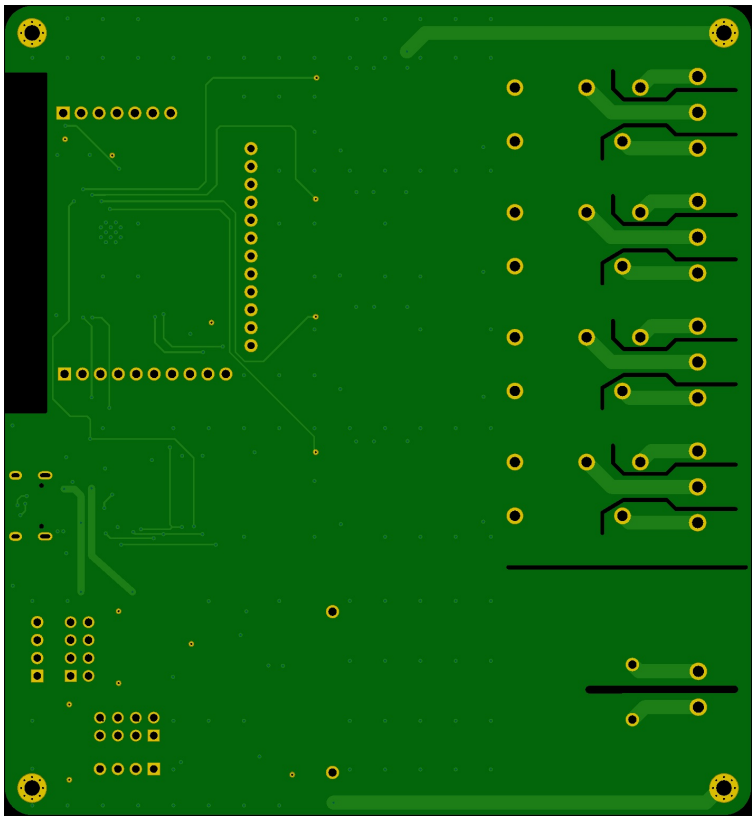
File name: 2024-06-05

Time: 2024-06-05 Layer count:4 PCB Thickness: 1.60 Quantity: 5 mm



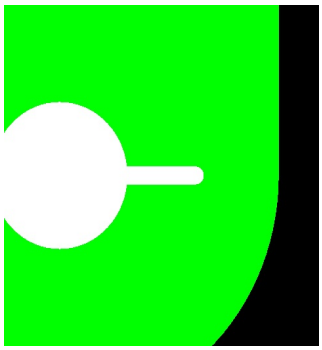
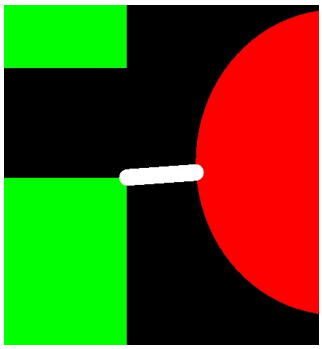
Basic Board Specs	Trace Width/Spacing	8.00/6.00mil
	Milling Density	40.5822m/m²
	Surface Finish Area	14.93%
	Test Point Count	399

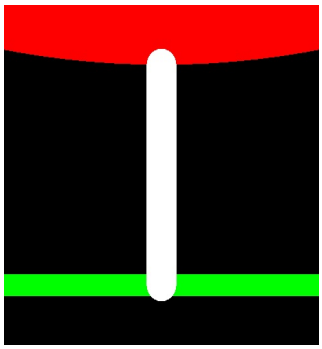
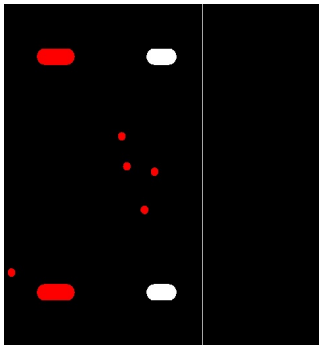


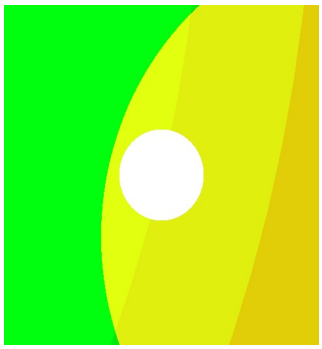
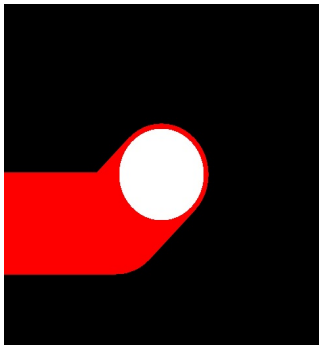


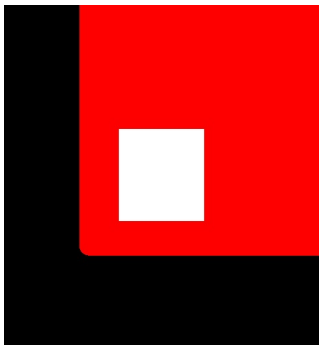
Type	Category	No. of Checks	Result
PCB Trace Analysis	Open/Shorts (IPC)	1	Fail
	Signal Integrity	4	Pass
	Smallest Trace Width	1	Pass 3
	Smallest Trace Spacing	3	Pass 161
	SMD Pad Spacing	1	Pass
	Pad Size	3	Pass 30
	Hatched Copper Pour	2	Pass
	Annular Ring Size	2	Pass 6 , Fail 1
	Drill to Copper	5	Pass 630 , Fail 2
	Copper-to-Board Edge	2	Pass 132 , Fail 34
	Holes on SMD Pads	4	Pass
PCB Drilling Analysis	Drill Diameter	8	Pass 41 , Fail 1
	Drill Spacing	4	Pass 18
	Drill to Board Edge	4	Pass
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
	Drill Hole Errors	3	Pass
PCB Solder Mask Analysis	Solder Mask Dam	2	Pass 12
	Missing SMask Opening	1	Pass
	Solder Paste Area	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Pass 5 , Fail 14

PCBA Component Analysis	Component Spacing	1	Fail
	Comp.-to-Board-Edge	3	Fail
	Componet Silkscreen Spacing	0	Fail
	Pad Count Mismatch	2	Fail
	Designator Length	0	Fail
	Double-sided Components	1	Pass
	ComponentClearanceAnalysi	1	Pass
PCBA Pin Analysis	Pin-to-SMD Pad	7	Fail
	Through-hole Pins	9	Fail
	Pressfit Pins	4	Fail
PCBA Pad Analysis	Chip Pad	60	Fail
	Pad-Trace Connections	4	Pass 302 , Fail 44
PCBA Fiducial Analysis	Fiducial Count	1	Fail
	Fiducial Analysis	3	Pass

ID	Check	Limits	Value	Issue	Image	Position	Qty	Level
1	Annular Ring Size_PTH Annular Ring	6,7,8	0.15 mm	PTH annular rings 6.10mil mil in size were detected in your design. This could increase the risk of tangency or breakouts, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended that PTH Pad Rings have a width of at least 6 mil.		96.98,-109.87	1	Warning
2	Drill to Copper_NPTH-to-Copper	8,10,12	0.17 mm	The NPTH to copper spacing should be at least 8 mil (ideally 12 mil). Spacing less than this could increase the risk of defects such as exposed copper, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. The NPTH to copper spacing in your design is only 6.75mil. It is recommended to increase the spacing to at least 12 mil.		100.26,-73.95	1	Risk

3	Copper-to-Board Edge_Copper-to-Board Edge	8,15,20	0.26 mm	Copper-to-edge spacing of 10.10mil was detected in your design.This could increase the risk of exposed copper on the edge of the boards or damaged traces/pads, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended to increase the spacing to at least 0.4 mm for edge routing and for v-cuts (v-cut spacing may depends on board thickness).		15.82,-30.70	4	Warning
4	Drill Diameter_Slot Aspect Ratio	12,10,8	0.31 mm	Slots with aspect ratio of 2.00 were detected in your design. This could increase the risk of incomplete drilling of the slot, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. The ratio should be increased to at least 2:1		104.37,-71.03	1	Warning

5	Silkscreen Spacing_Solder Mask-to-Silkscreen	4,5,6	0.00 mm	For most factories, the minimum silkscreen to solder mask spacing requirement is at least 8 mil. Failure to meet the factory's requirements could result in part of the silkscreen being removed or being printed directly on the pads, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. Silkscreen to solder mask spacing of 0 mil were detected in your design. It is recommended to increase the spacing to at least 12 mil.		105.26,-75.50	1	Risk
6	Pad-Trace Connections _THT Pad-Trace Width	-,-,-	Not analyzed	If the trace width is greater than 100.00% of the width of the solder pad, there will be increased thermal dissipation to the connected copper during soldering that may increase solder defects such as cold solder joints. It is recommended to reduce the trace width near the pad (necking down).		16.95,-93.53	2	Risk

7	Pad-Trace Connections _SMD Pad-Trace Width	-,-,-	Not analyzed	If the trace width is greater than 100.00% of the width of the solder pad, there will be increased thermal dissipation to the connected copper during soldering that may increase solder defects such as cold solder joints. It is recommended to reduce the trace width near the pad (necking down).		89.47,-33.64	35	Risk