

HQDFM Design for Manufacture(DFM) Report

File name: 2024-12-04

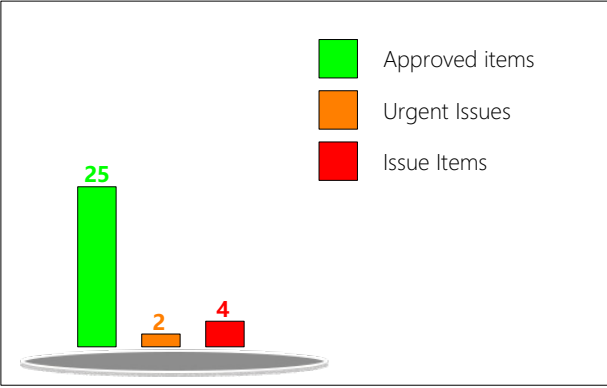
Time: 2024-12-04

Layer count:4

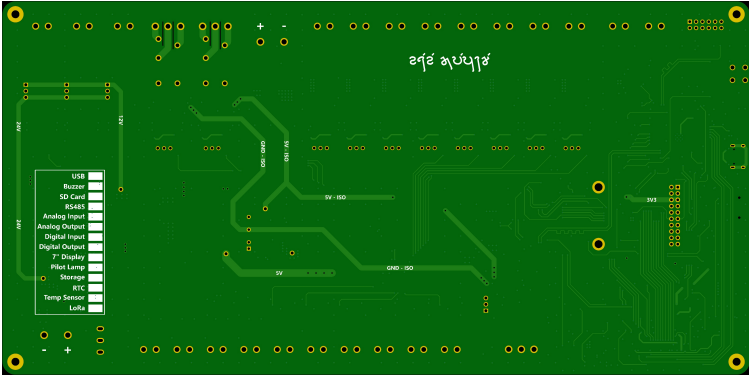
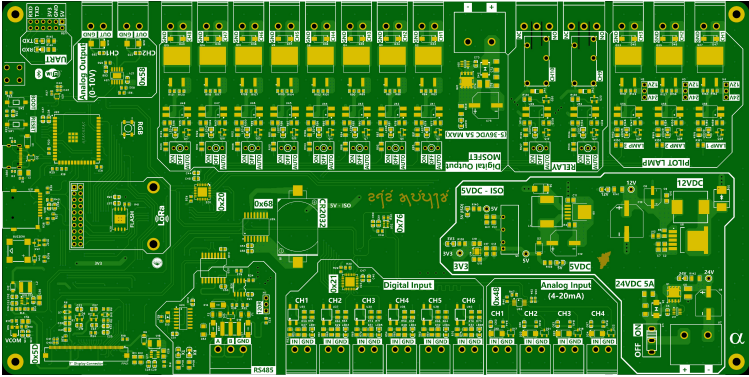
PCB Thickness: 1.60

Quantity: 5

mm

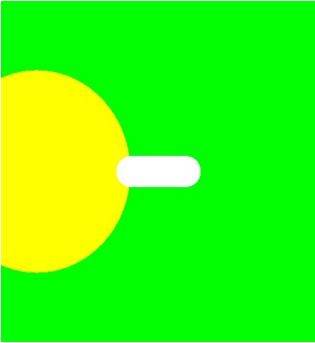
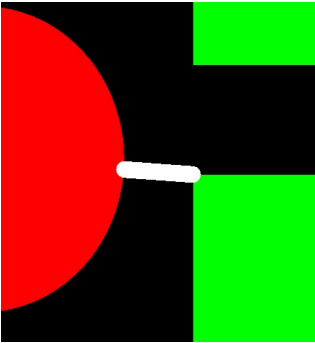


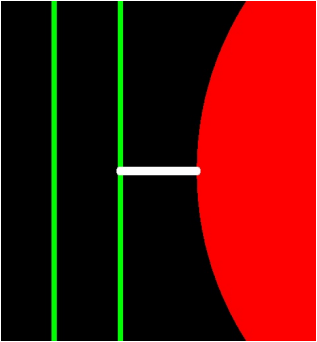
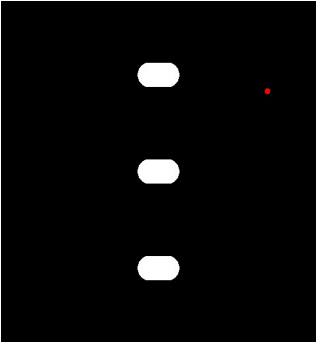
Basic Board Specs	Trace Width/Spacing	10.00/6.00mil
	Milling Density	19.9510m/m²
	Surface Finish Area	15.57%
	Test Point Count	1385

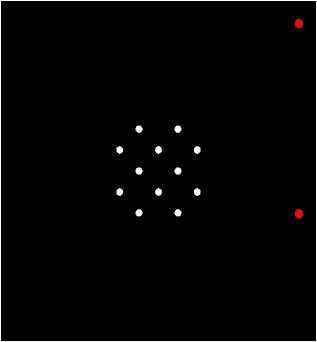
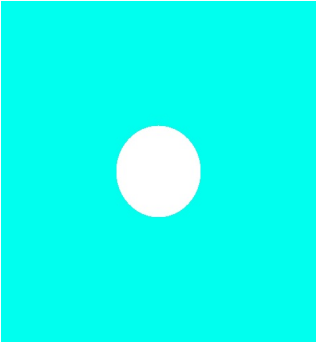


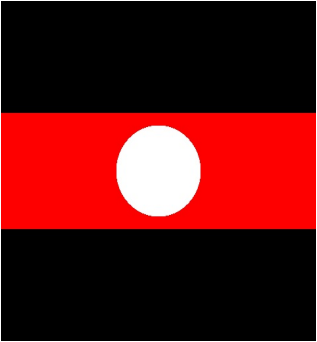
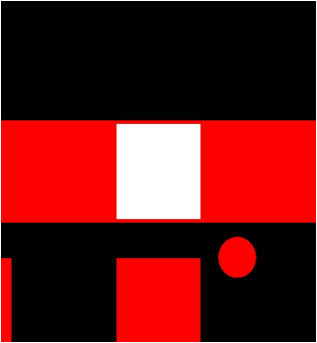
Type	Category	No. of Checks	Result
PCB Trace Analysis	Open/Shorts (IPC)	1	Fail
	Signal Integrity	4	Pass
	Smallest Trace Width	1	Pass 2
	Smallest Trace Spacing	3	Pass 371
	SMD Pad Spacing	1	Pass
	Pad Size	3	Pass 59
	Hatched Copper Pour	2	Pass
	Annular Ring Size	2	Pass 4 , Fail 2
	Drill to Copper	5	Pass 3102 , Fail 2
	Copper-to-Board Edge	2	Pass 58 , Fail 16
	Holes on SMD Pads	4	Pass
PCB Drilling Analysis	Drill Diameter	8	Pass 57 , Fail 4
	Drill Spacing	4	Pass 114
	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 6
	Missing SMask Opening	1	Pass
	Solder Paste Area	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Pass 86 , Fail 15

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
	ComponentClearanceAnalysis	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 1014 , Fail 278
PCBA Fiducial Analysis	Fiducial Count	1	Fail
	Fiducial Analysis	3	Pass

ID	Check	Limits	Value	Issue	Image	Position	Qty	Level
1	Annular Ring Size_Via Annular Ring	6,7,8	0.08 mm	Min via annular rings2.95mil in size were detected in your design.It will affect production efficiency and electrical reliability. It is recommended that the minimum ring size for "via annular rings" be ≥5 mils.		28.63,-54.01	1	Risk
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 6.74mil. It is recommended to increase the spacing to at least 12 mil.		6.83,-65.30	1	Risk

3	Copper-to-Board Edge_Copper-to-Board Edge	8,15,20	0.35 mm	<p>Copper-to-edge spacing of 13.88mil 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).</p>		216.84,-15.24	7	Warning
4	Drill Diameter_Slot Aspect Ratio	12,10,8	0.25 mm	<p>Slots with aspect ratio of 1.83 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</p>		260.60,-136.14	1	Warning

5	Drill Diameter_Smallest Drill Size	12,10,8	0.25 mm	<p>Min.Drill:0.25mm in diameter were detected in your design. Smaller drill bits necessitate more frequent replacements, increasing the likelihood of missed holes, misalignment, and rough hole walls. This diminishes manufacturing efficiency, reduces yield, and impacts board reliability. Holes with diameters of 0.1mm or less require laser drilling and must adhere to stringent board width requirements. It is advisable to increase the diameter to at least 0.2mm or 0.3mm to avoid additional costs</p>		28.47,-54.71	1	Warning
6	Silkscreen Spacing_Solder Mask-to-Silkscreen	4,5,6	0.00 mm	<p>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.</p>		13.23,-76.42	3	Risk

7	Pad-Trace Connections_THT Pad-Trace Width	-,-,-	Not analyzed	<p>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).</p>		201.04,-91.69	17	Risk
8	Pad-Trace Connections_SMD Pad-Trace Width	-,-,-	Not analyzed	<p>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).</p>		197.36,-99.30	239	Risk