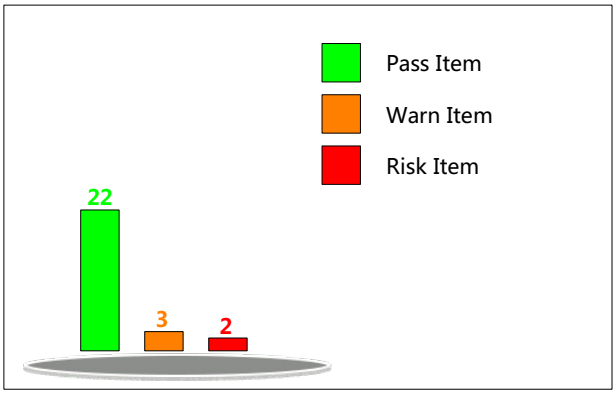
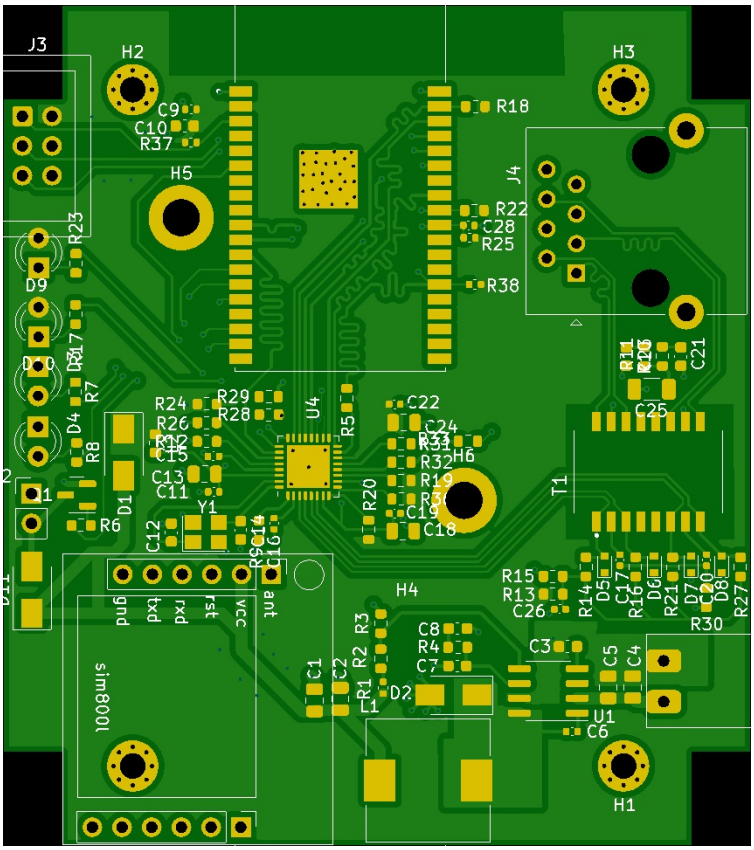
HQDFM Design for Manufacture(DFM) Report

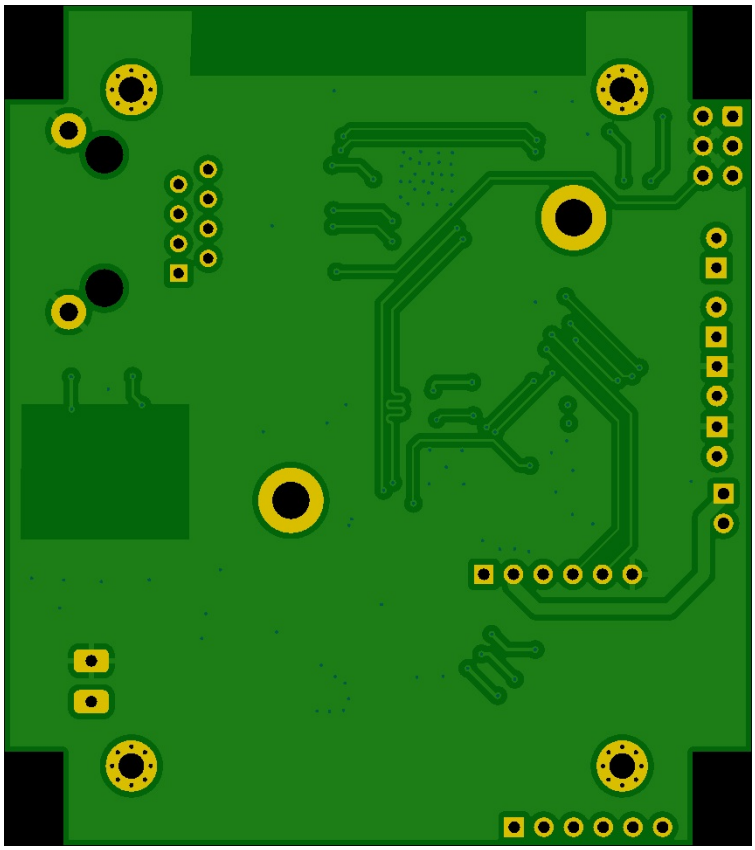
File name: 0000A61279_1

Time: 2024-02-08 Layer num:2 pcb thickness:1.60 quantity:5 Board Size:64.00*72.00 mm

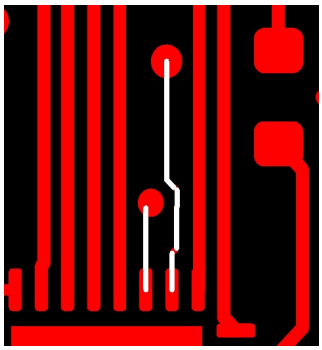
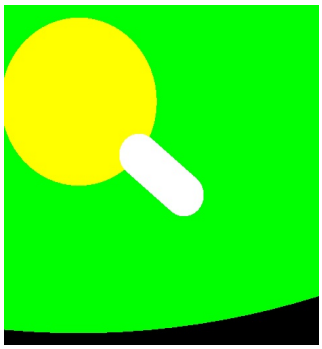


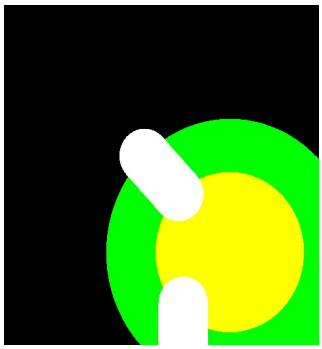
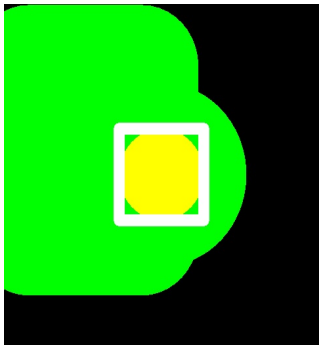
param_analyze	Trace Width/Spacing	4.00/8.00mil
	Milling Density	60.5500m/m²
	Surface Finish Area	21.52%
	Test Point Count	297
	Panel Efficiency	88.1948%

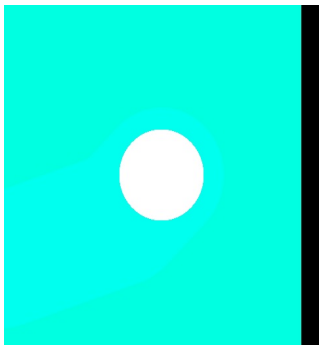




type	checkitem	checksubitem	result
pcb_signal	Smallest Trace Width	1	Pass 4 ,Fail 1
	Smallest Trace Spacing	3	Pass 352
	Pad Spacing	2	Pass 60
	Pad Size	3	Pass 231
	Hatched Copper Pour	2	Pass
	RingHole	2	Pass 244 ,Fail 66
	Drill to Copper	5	Pass 100
	Signal Integrity	4	Pass
	Board Edge Clearance	2	Pass 58
	Holes on SMD Pads	4	Fail
	Open/Shorts (IPC)	1	Fail
pcb_drill	Hole Diameter	8	Pass 42
	Drill Hole Density	1	Pass
	Hole Diameter	8	Pass 42
	Drill Hole Spacing	4	Pass 28
	Drill to Board Edge	4	Pass
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
pcb_soldmask	Solder Mask Spacing	2	Pass 60
	Missing SMask Openings	1	Pass
pcb_silk	Silkscreen Spacing	1	Pass 2 ,Fail 24
ass_markpoint	Fiducials	1	Fail

ID	item	rule	value	issue	image	Coordinate	count	level
1	Smallest Trace Width_Smallest Trace Width	3.5,4,5	0.10 mm	Traces 3.94mil in width were detected in your design. This could result in overetched traces, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended to increase the width to at least 6 mil for regular routing and at least 4 mil in high density areas, such as when routing fine pitch BGAs.		69.18,-113.03	1	Warn
2	RingHole_PTH Annular Ring	6,7,8	3.89 mil	Your plug-in hole welding ring is5.87,which will affect production efficiency and electrical reliability. It is suggested that the minimum ring of plug-in hole welding ring is >= 8 mil		53.31,-87.08	64	Risk

3	RingHole_Via Annular Ring	6,7,8	3.89 mil	Your through hole ring is3.89,which will affect production efficiency and electrical reliability. It is suggested that the minimum ring of through hole ring is >= 5 mil		68.84,-113.37	2	Risk
4	Holes on SMD Pads_Via on SMD Pad	-,-,-	93.59%	Holes on surface mount pads were detected in your design. During SMT assembly, solder could leak into the hole and pull solder away from the SMD contact, which could decrease manufacturing efficiency and yield, and affects the reliability of the boards. Please check and separate the holes from the pads if possible.		65.84,-113.15	1	Warn

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.		96.81,-113.10	24	Risk