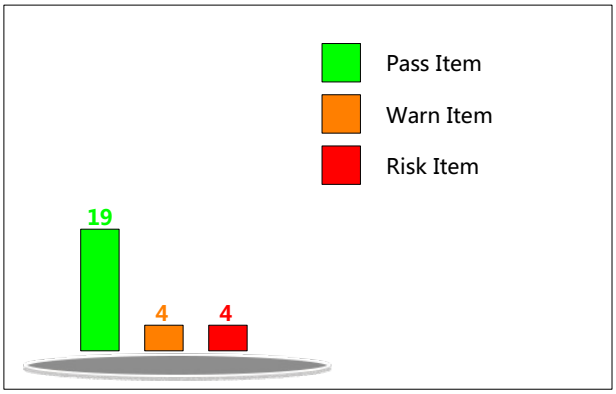
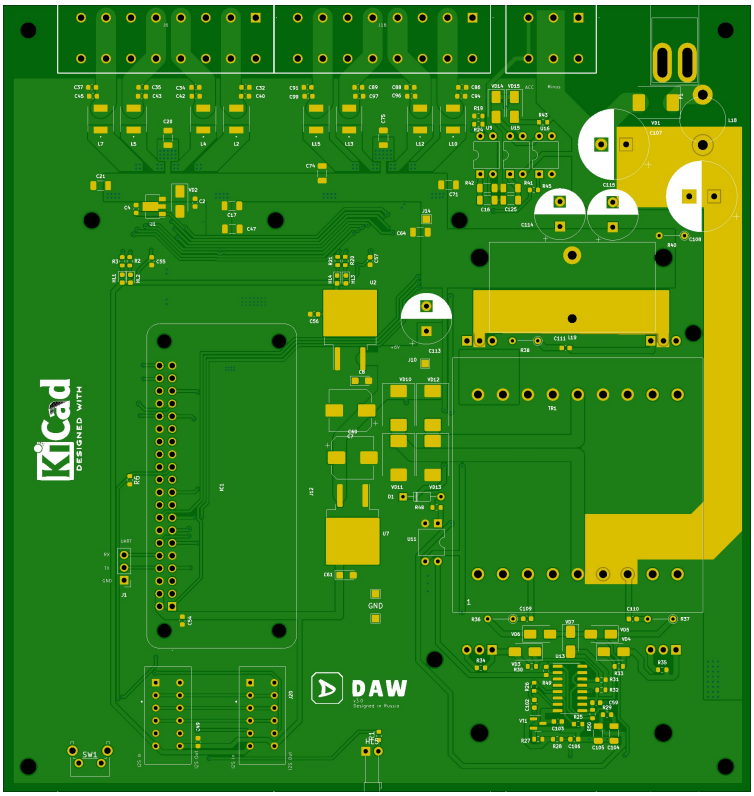
HQDFM Design for Manufacture(DFM) Report

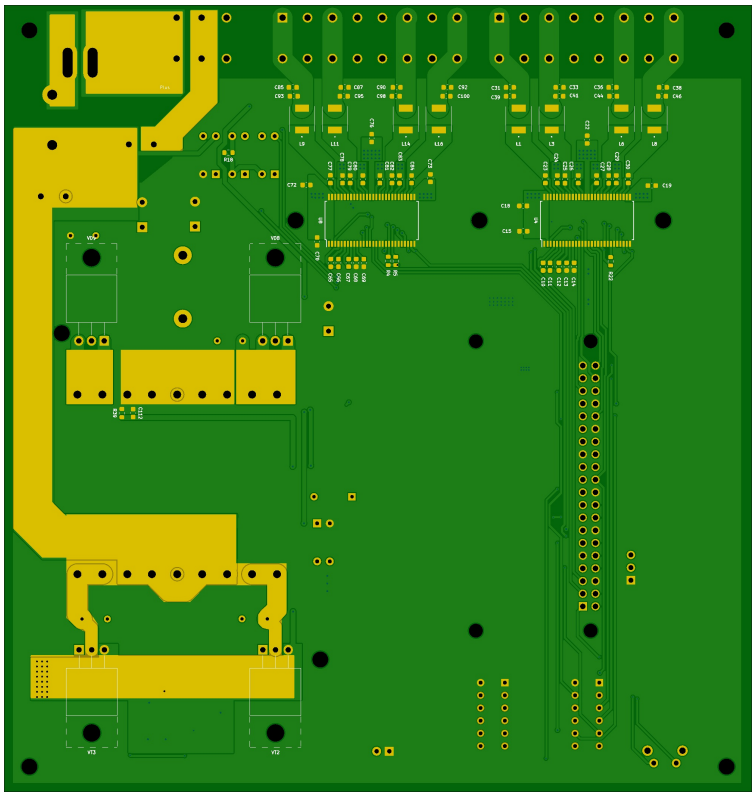
File name: 0000A190374_1

Time: 2024-08-21 Layer num:2 pcb thickness:1.60 quantity:5 Board Size:149.86*157.73 mm

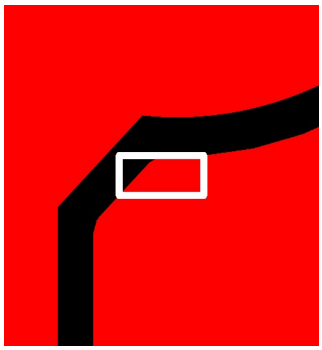
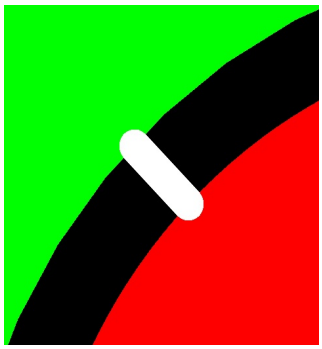


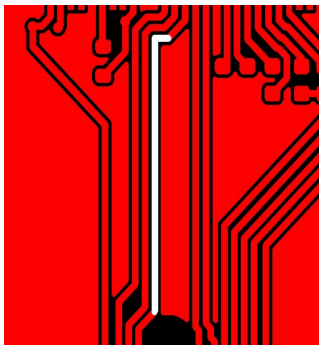
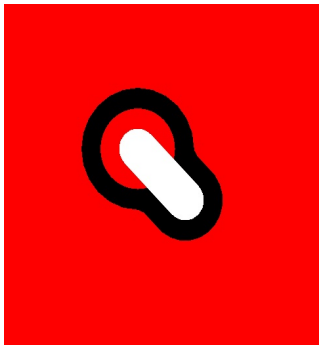
param_analyze	Trace Width/Spacing	10.00/7.00mil+
	Milling Density	26.3672m/m²
	Surface Finish Area	30.05%
	Test Point Count	683
	Panel Efficiency	87.6998%

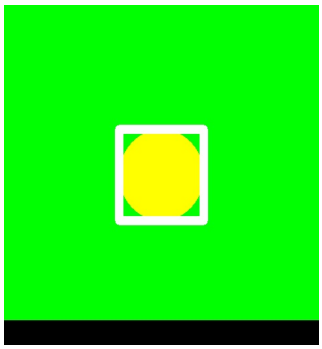
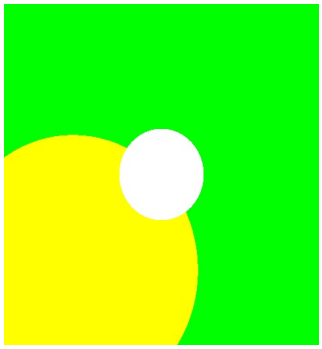


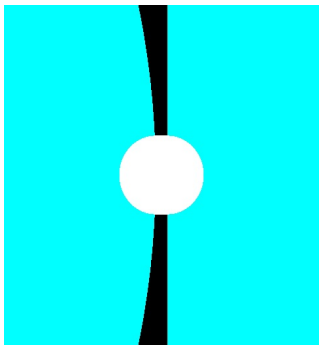
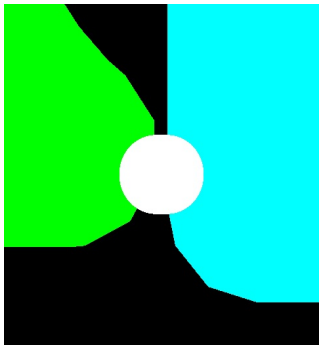


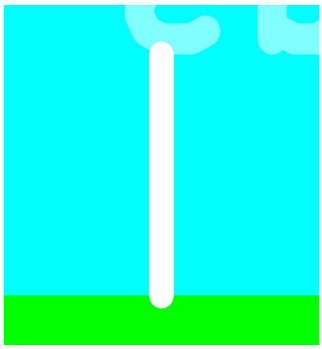
type	checkitem	checksubitem	result
pcb_signal	Smallest Trace Width	1	Pass
	Smallest Trace Spacing	3	Pass 2250
	Pad Spacing	2	Pass 110
	Pad Size	3	Pass 362 ,Fail 2
	Hatched Copper Pour	2	Pass
	RingHole	2	Pass 429
	Drill to Copper	5	Pass 431 ,Fail 30
	Signal Integrity	4	Fail 5
	Board Edge Clearance	2	Pass 16
	Holes on SMD Pads	4	Fail 4
	Open/Shorts (IPC)	1	Fail
pcb_drill	Hole Diameter	8	Pass 98
	Drill Hole Density	1	Pass
	Hole Diameter	8	Pass 98
	Drill Hole Spacing	4	Pass 134 ,Fail 8
	Drill to Board Edge	4	Pass
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
pcb_soldmask	Solder Mask Spacing	2	Pass 155 ,Fail 7
	Missing SMask Openings	1	Pass
pcb_silk	Silkscreen Spacing	1	Pass 5 ,Fail 12
ass_markpoint	Fiducials	1	Fail

ID	item	rule	value	issue	image	Coordinate	count	level
1	Pad Size_Long Pads	8,14,22	0.21 mm	Rectangular/Oblong pads of width 0.21mm were detected in your design. This could result in overetching, detached pads and increased testing costs, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. The pad width should be enlarged to at least 0.2 mm if possible.		134.15,-125.37	2	Warn
2	Drill to Copper_NPTH-to-Copper	8,10,12	0.25 mm	mil. It is recommended to increase the spacing to at least 12 mil.9.83milIt is recommended to increase the spacing to at least 12 mil.		94.14,-144.57	1	Warn

3	Signal Integrity_Acute Angle Traces	-,-,-	Error(s) detected	Acute angle traces were detected in your design. The sharp change in direction could cause signal reflections, resulting in transmission line and signal integrity issues. Consider increasing the angle or rounding the traces.		31.88,-57.82	1	Risk
4	Signal Integrity_Unconnected Traces	-,-,-	Error(s) detected	Unconnected traces or dangling lines were detected in your design. This could be the result of human error and can affect the functionality or performance of the boards. Please investigate and modify the design if necessary. Unconnected traces less than 6 mil in length should be removed.		81.03,-79.88	1	Risk

5	Holes on SMD Pads_Via on SMD Pad	-,-,-	100.00 %	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.		142.24,-138.68	4	Warn
6	Drill Hole Spacing_Same Net Via Spacing	0.15,0.15,0.2	0.00 mm	For most factories, the same net via spacing requirement is 6-8 mil. Failure to meet the factory's requirements could increase the risk of the drill bit breaking, resulting in more drill hole defects, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. Same net via spacing of 0.00mil were detected in your design. It is recommended to increase the spacing to at least 8 mil.		111.99,-145.31	1	Risk

7	Solder Mask Spacing_Solder Mask Dam	5,5.5,6	0.63 mil	Solder mask dam of width less than 0.63mil 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. The width should be at least 6 mil where possible.		123.18,-39.53	5	Risk
8	Solder Mask Spacing_Solder Mask-to-Trace	5,5.5,6	0.02 mm	Solder mask opening to trace spacings less than 0.63mil were detected in your design. This could increase the risk of exposed traces and short circuits, which decrease manufacturing efficiency and yield. If the value is negative, exposed traces were detected in the design which are susceptible to shorts during assembly. The width should be increased to at least 0.08mm.		114.04,-29.05	1	Risk

9	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.		116.82,-17.32	1	Risk