

HQDFM Design for Manufacture(DFM) Report

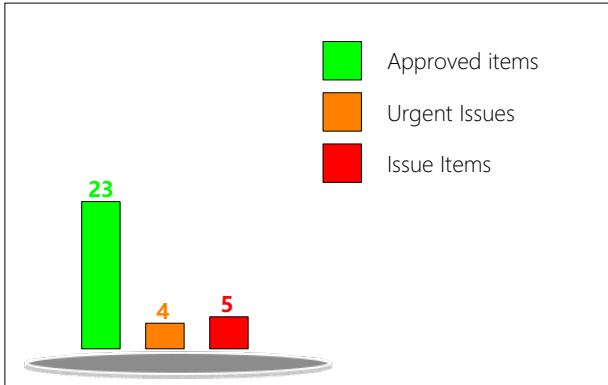
File name: GERBER

Time: 2025-06-28Layer count:4

PCB Thickness: 1.60

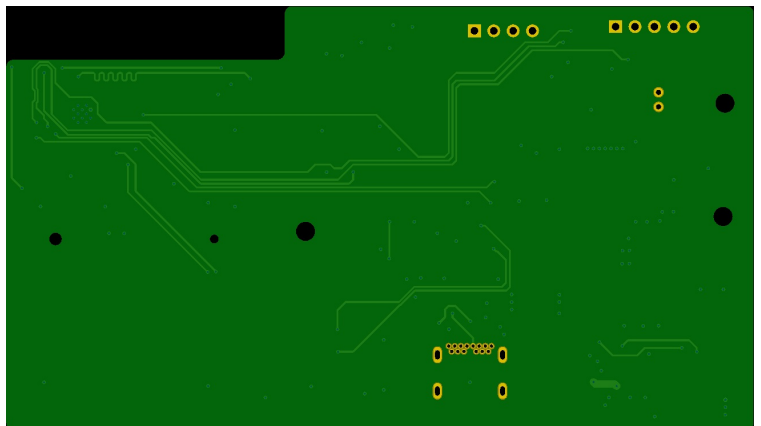
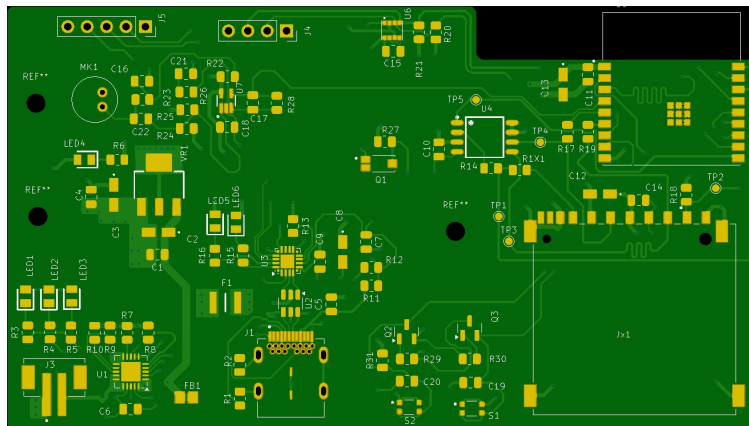
Quantity: 5

mm



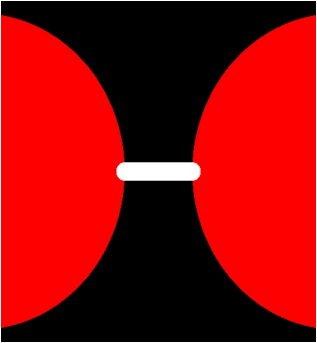
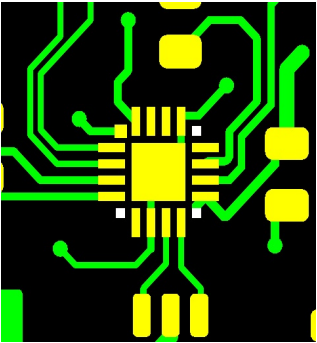
Basic Board Specs	Trace Width/Spacing	8.00/6.00mil
	Milling Density	57.2496m/m ²
	Surface Finish Area	9.02%
	Test Point Count	318

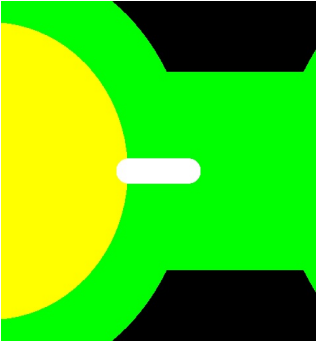
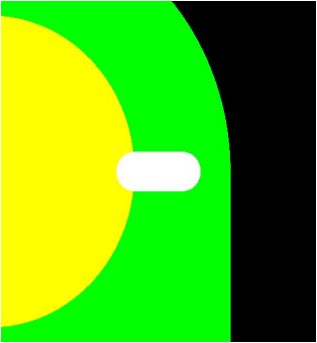
The file size is small, which can affect the surface mount assembly process. It is recommended to have a size larger than 7*7cm. You can optimize the size by adding a process edge or increasing the panelization.

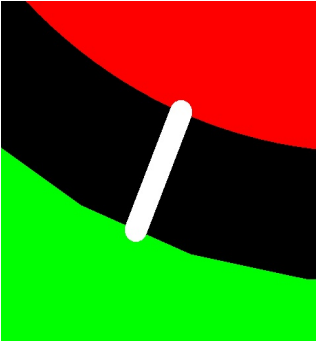
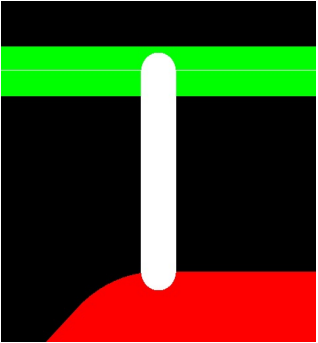


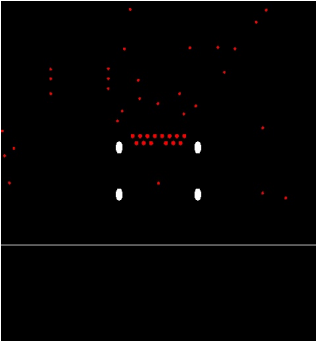
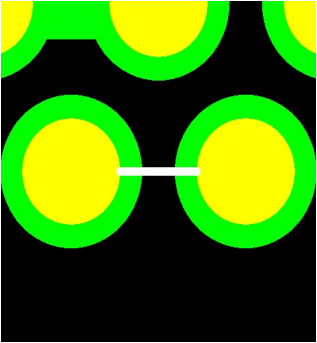
Type	Category	No. of Checks	Result
PCB Trace Analysis	Open/Shorts (IPC)	1	Fail
	Signal Integrity	4	Pass
	Smallest Trace Width	1	Pass 5
	Smallest Trace Spacing	3	Pass 491 , Fail 38
	SMD Pad Spacing	1	Pass
	Pad Size	3	Pass 31 , Fail 1
	Hatched Copper Pour	2	Pass
	Annular Ring Size	2	Pass 3 , Fail 3
	Drill to Copper	5	Pass 262 , Fail 10
	Copper-to-Board Edge	2	Pass 20 , Fail 26
	Holes on SMD Pads	4	Pass
PCB Drilling Analysis	Drill Diameter	8	Pass 15 , Fail 1
	Drill Spacing	4	Pass 25 , Fail 11
	Drill to Board Edge	4	Pass 9
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
	Drill Hole Errors	3	Pass
PCB Solder Mask Analysis	Solder Mask Dam	3	Pass 20 , Fail 4
	Missing SMask Opening	1	Pass
	Solder Paste Area	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Pass 5

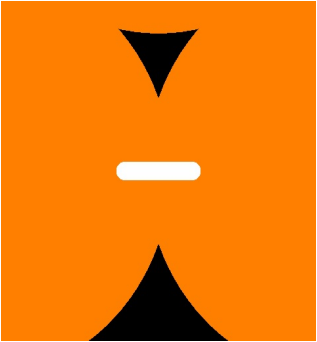
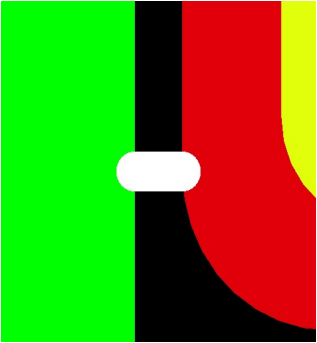
PCBA Component Analysis	Component Spacing	2	Fail
	Comp Height Analysis	3	Pass
	Comp.-to-Board-Edge	3	Fail
	Component Silkscreen Spacing	0	Fail
	Pad Count Mismatch	4	Fail
	Designator Length	0	Fail
	Double-sided Components	1	Pass
	Mid Mount Comp	1	Pass
PCBA Pin Analysis	SMT Pad Analysis	8	Fail
	Through-hole Pins	10	Fail
PCBA Pad Analysis	Chip Pads	60	Fail
PCBA Fiducial Analysis	Fiducial Count	1	Fail
	Fiducial Analysis	4	Pass

ID	Check	Limits	Value	Issue	Image	Position	Qty	Level
1	Smallest Trace Spacing_SMD Pad Spacing	4,6,10	0.15 mm	<p>Pad-to-pad spacing of 5.91mil were detected in your design. This could result in incomplete solder mask dams between the pads, thereby increasing the risk of incomplete etching of the pads, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended to increase the spacing to at least 6 mill. For solder mask covered pads, 4 mil spacing is sufficient.</p>		184.80,-129.26	21	Warning
2	Pad Size_Short Pads	8,12,20	0.30 mm	<p>Regular pads of size 0.30mm 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 pads should be enlarged to at least 0.3 mm if possible.</p>		182.75,-118.15	15	Warning

3	Annular Ring Size_PTH Annular Ring	6,7,8	0.05 mm	<p>PTH Pad Rings3.94 mil in size were detected in your design. This could increase the risk of tangency or breakouts, which decrease electrical reliability and yield, and affect the reliability of the boards. It is recommended that PTH Pad Rings have a width of at least 6 mil.</p>		180.68,-129.26	2	Risk
4	Annular Ring Size_Via Annular Ring	4,5,6	0.05 mm	<p>Min via annular rings1.97mil 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.</p>		233.08,-99.42	4	Risk

5	Drill to Copper_NPTH-to-Copper	8,10,12	0.20 mm	<p>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.</p> <p>The NPTH to copper spacing in your design is only 7.89mil. It is recommended to increase the spacing to at least 12 mil.</p>		237.15,-116.10	11	Risk
6	Copper-to-Board Edge_Copper-to-Board Edge	8,11,15	0.20 mm	<p>For most factories, the copper-to-board edge clearance requirement is 0.2 mm. Failure to meet the factory's requirements 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.</p> <p>Copper-to-board edge spacing of 7.89mil was detected in your design. It is recommended to increase the spacing to at least 0.25 mm for edge routing and 0.4 mm for v-cuts (v-cut spacing may depends on board thickness).</p>		238.42,-91.85	4	Risk

7	Drill Diameter_Slot Aspect Ratio	1.5,2,2	0.30 mm	<p>Slots with aspect ratio of 1.92 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>		183.20,-132.78	1	Warning
8	Drill Spacing_Different Net PTH Spacing	0.4,0.45,0.5	0.35 mm	<p>For most factories, the different net PTH spacing requirement is 16-18 mil. Failure to meet the factory's requirements could increase the risk of solder bridges during soldering, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. Different net PTH spacing of 13.78mil were detected in your design. It is recommended to increase the spacing to at least 17 mil.</p>		181.20,-129.97	11	Risk

9	Solder Mask Dam_Solder Mask Dam	5,6,10	1.93 mil	<p>Solder mask dam of width less than 5.91mil were detected in your design.</p> <p>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.</p> <p>The width should be at least 6 mil where possible.</p>		182.00,-129.97	2	Warning
10	Solder Mask Dam_Solder Mask-to-Trace	1.5,2,2.5	0.05 mm	<p>Solder mask opening to trace spacings less than 1.93mil were detected in your design. This could increase the risk of exposed traces and short circuits, which decrease manufacturing efficiency and yield. The width should be increased to at least 0.08mm.</p>		175.05,-96.75	9	Warning