

HQDFM Design for Manufacture(DFM) Report

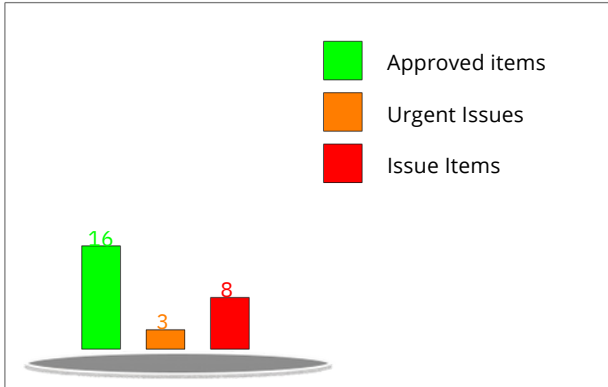
File name: 0000A542351_1

Time: 2025-09-25Layer count:4

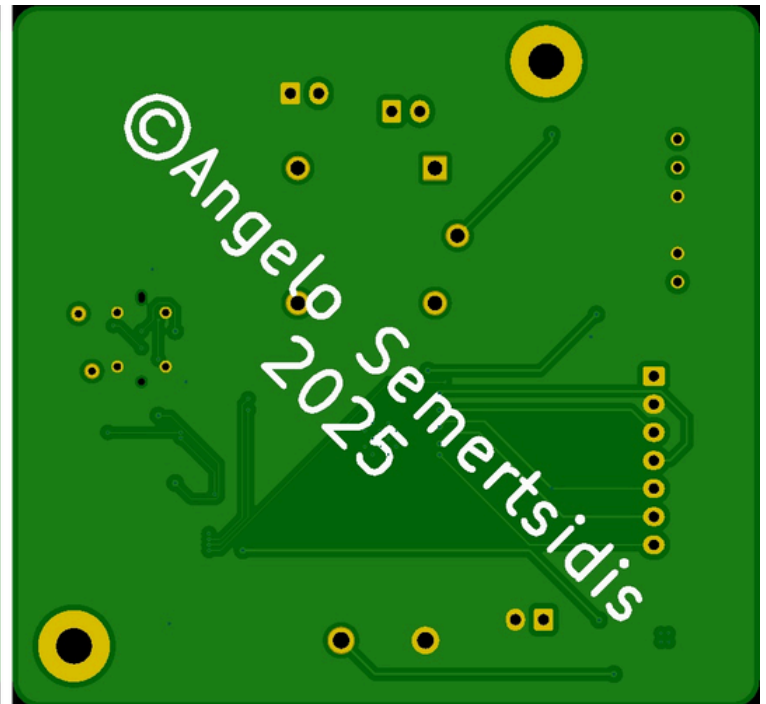
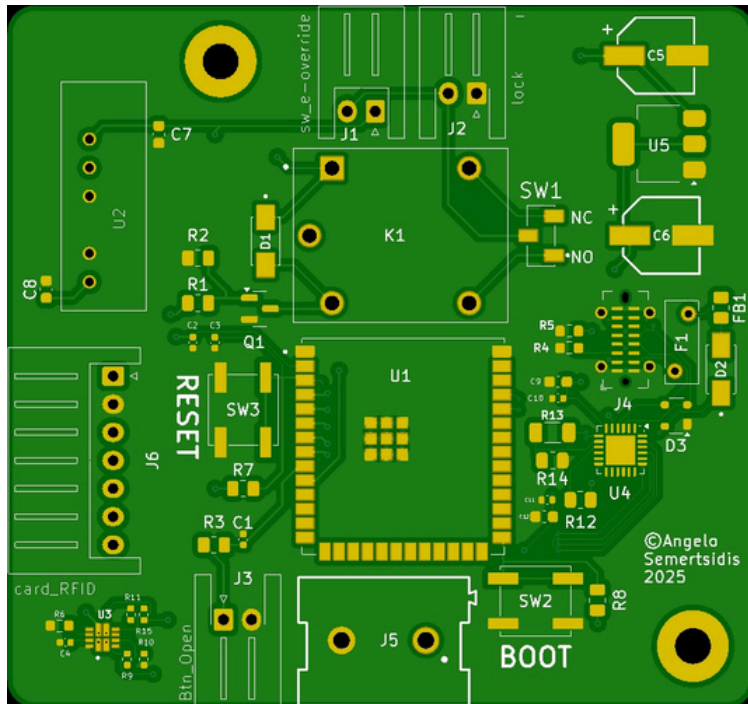
PCB Thickness: 1.60

Quantity: 5

mm

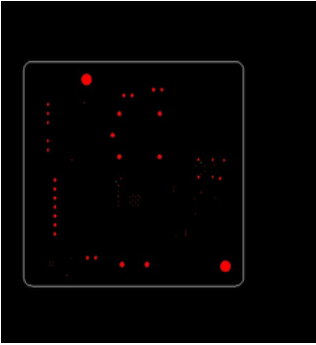
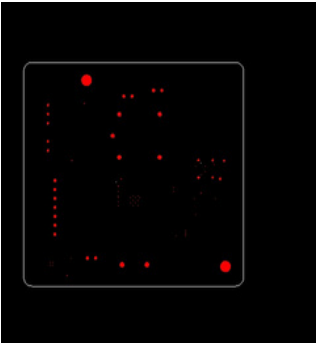


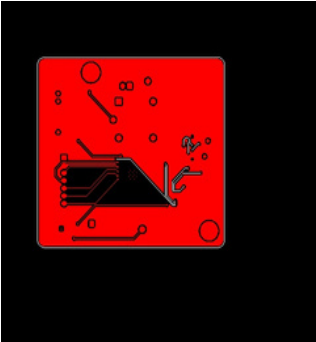

Basic Board Specs	Trace Width/Spacing	4.00/8.00mil
	Milling Density	62.7329m/m²
	Surface Finish Area	14.30%
	Test Point Count	221

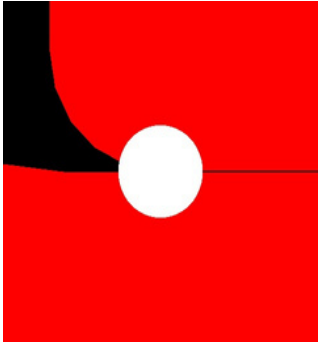
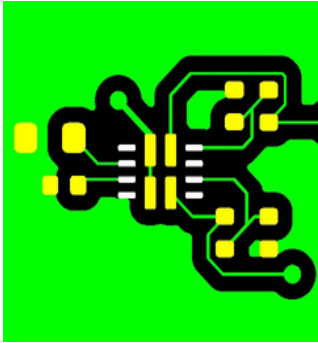


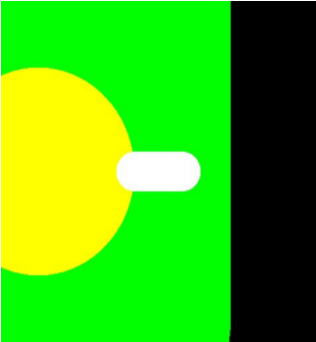
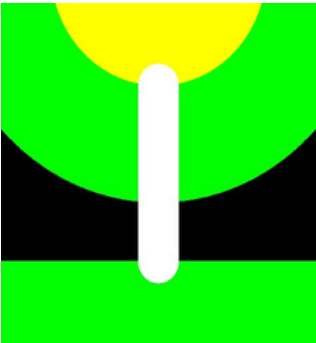
PASS
WITH ERRORS

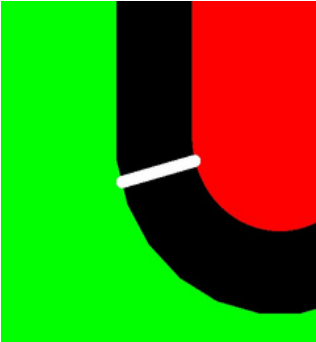
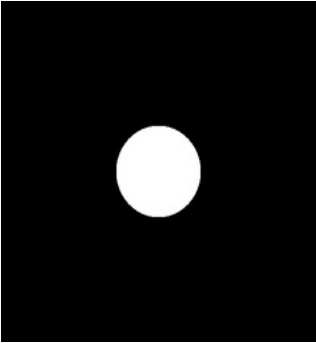
Type	Category	No. of Checks	Result
PCB Trace Analysis	Smallest Trace Width	1	Pass 2 , Fail 2
	Smallest Trace Spacing	3	Pass 149 , Fail 41
	SMD Pad Spacing	2	Pass 66
	Pad Size	3	Pass 23 , Fail 1
	Hatched Copper Pour	2	Pass
	Annular Ring Size	2	Pass 4 , Fail 4
	Drill to Copper	5	Pass 178 , Fail 26
	Signal Integrity	4	Fail 12
	Copper-to-Board Edge	2	Pass 32
	Holes on SMD Pads	4	Fail 4
	Open/Shorts (IPC)	1	Fail
PCB Drilling Analysis	Drill Diameter	8	Pass 78 , Fail 6
	Drill Hole Density	1	Pass
	Drill Diameter	8	Pass 78 , Fail 6
	Drill Spacing	4	Pass 3
	Drill to Board Edge	4	Pass
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
PCB Solder Mask Analysis	Solder Mask Dam	2	Pass 8 , Fail 25
	Missing SMask Opening	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Fail
PCBA Fiducial Analysis	Fiducial Count	1	Pass

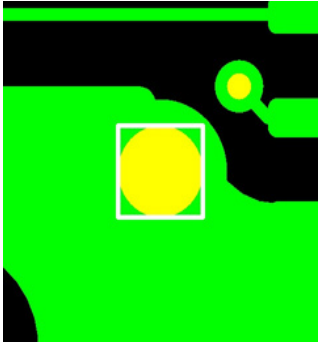
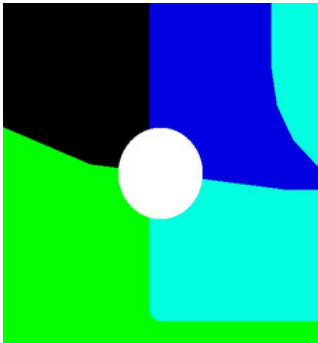
ID	Check	Limits	Value	Issue	Image	Position	Qty	Level
1	Drill Diameter_Aspect Ratio	12,10,8	0.10 mm	For most factories, the aspect ratio is 8:1 to 10:1 (Depth of hole/Diameter of hole). Smaller aspect ratios require higher throwing power to ensure the interior of the holes are fully plated. Failure to meet the factory's requirements could increase the risk of incomplete plating of the hole, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. Aspect ratio of 16.00was detected in your design. The drill sizes and board thickness should be modified to reduce the aspect ratio to at least 8:1.		152.25,-97.75	24	Risk
2	Drill Diameter_Smallest Drill Size	0, 2,0.25,0.3 mm	0.10 mm	For most factories, the smallest (mechanical) drill hole size requirement is around 0.2-0.25 mm. Smaller drill bits need to be replaced frequently and are more likely to result in missing holes, misalignment and hole wall roughness, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. 0.1 mm and smaller drill holes require laser drilling and have strict board width requirements. Drill holes 0.10mm in diameter were detected in your design. It is advisable to increase the diameter to at least 0.2mm or 0.3mm to avoid additional costs		152.25,-97.75	24	Risk

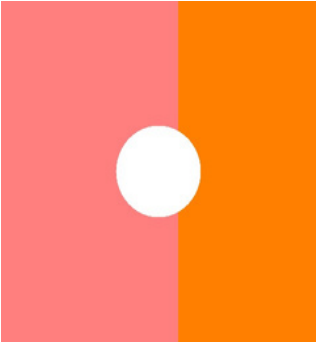
3	Smallest Trace Width_Smallest Trace Width	3.5,4,5	0.10 mm	<p>Traces 3.94mil in width were detected in your design. This could result in overetched traces, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended to increase the width to at least 6 mil for regular routing and at least 4 mil in high density areas, such as when routing fine pitch BGAs.</p>		154.50,-104.50	2	Warning
4	Smallest Trace Spacing_SMD Pad Spacing	3.5,4,5	0.02 mil	<p>For most factories, the minimum pad-to-pad spacing requirement is around 3-4 mil. Failure to meet the factory's requirement 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. Pad-to-pad spacings of 1.97mil were detected in your design. The spacing should be at least 6 mil where possible. For solder mask covered pads, 4 mil spacing is sufficient.</p>		160.50,-114.25	6	Risk

5	Smallest Trace Spacing_Trace-to-Pad Spacing	3.5,4,5	0.02 mil	<p>For most factories, the minimum trace-to-pad spacing requirement is around 3.0-4.0 mil, with 3.5 mil or less requiring tighter quality control. Failure to meet the factory's requirement could result in insufficient solder mask covering the trace or excess solder mask covering the pad, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. Trace-to-pad spacings of 0.02mil were detected in your design. The spacing should be at least 4 mil where possible.</p>		118.58,-124.10	44	Risk
6	Pad Size_Long Pads	7,9,10	0.20 mm	<p>Rectangular/Oblong pads of width 0.20mm 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 0.25 mm if possible.</p>		119.98,-123.25	11	Warning

7	Annular Ring Size_Via Annular Ring	4,5,6	0.05 mm	<p>Min via annular rings 1.97 mil 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>		119.75,-122.85	5	Risk
8	Drill to Copper_Via-to-Trace [Outer]	8,10,12	5.91 mil	<p>For most factories, the via to trace clearance requirement in outer copper layers is at least 8 mil. Failure to meet the factory's requirement could increase the risk of defects such as short circuits, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. Via to trace spacing (outer layer) of 5.91 mil was detected in your design. The spacing should be at least 10 mil.</p>		143.85,-106.44	76	Risk

9	Drill to Copper_NP TH-to-Copper	8,10,12	0.15 mm	<p>NPTH to copper spacing of 9.86mil was detected in your design. This could increase the risk of defects such as short circuits, which decrease manufacturing efficiency and yield, and affect the reliability of the boards. It is recommended to increase the spacing to at least 12 mil.</p>		166.10,-93.32	8	Risk
10	Signal Integrity_Unconnected Vias	---	Error(s) detected	<p>Isolated areas of copper (floating copper) were detected in your design. Leaving isolated areas of copper unconnected can cause them to act as antennas and can affect signal integrity. We suggest removing or grounding them.</p>		143.85,-104.86	12	Risk

11	Holes on SMD Pads_PTH on SMD Pad	-,-,-	100.00 %	<p>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.</p>		164.35,-99.15	4	Warning
12	Solder Mask Dam_Solder Mask-to-Trace	1.5,2,2.5	0.00 mm	<p>Solder mask opening to trace spacings less than 0.00mil 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.</p>		118.48,-124.09	26	Risk

13	Silkscreen Spacing _Solder Mask-to- Silkscreen	4,5,6	Error(s) detect ed	<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>		151.89,-93.67	1	Risk
----	--	-------	--------------------------	--	--	---------------	---	------