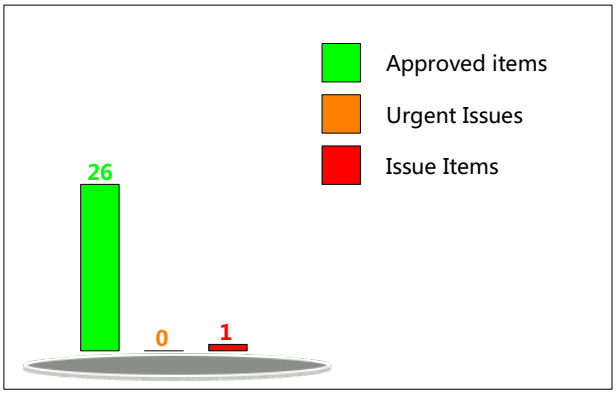
HQDFM Design for Manufacture(DFM) Report

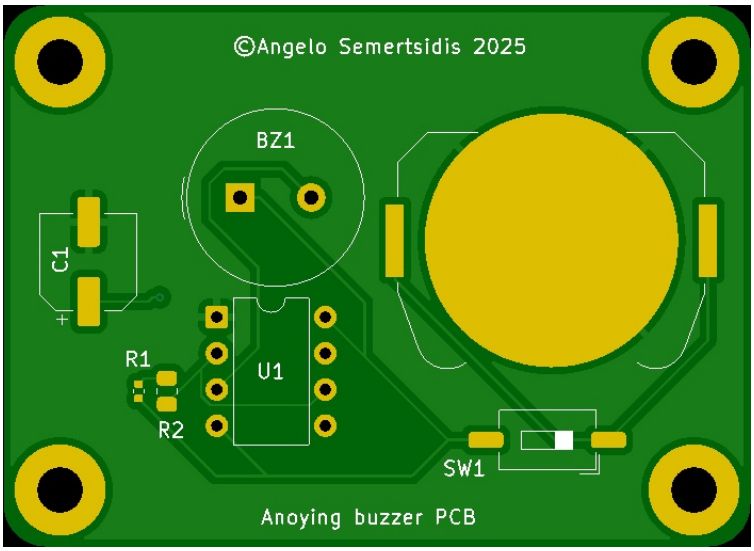
File name: 0000A546986_1

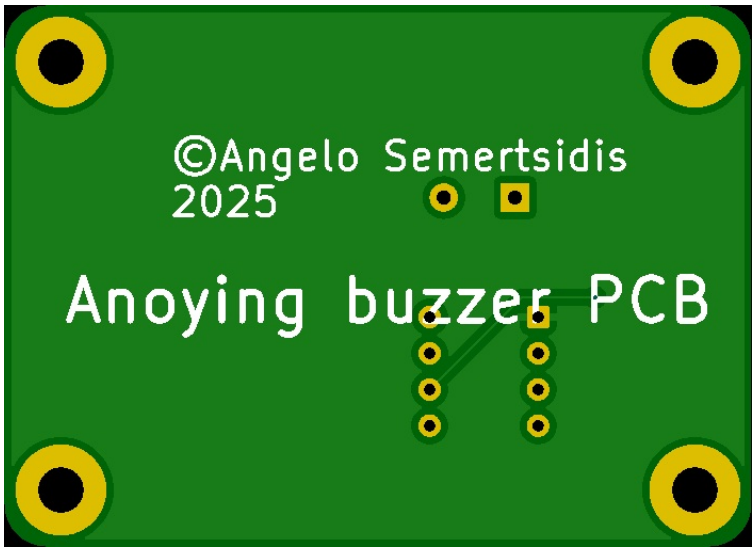
Time: 2025-09-30 Layer count:2 PCB Thickness: 1.60 Quantity: 5 mm



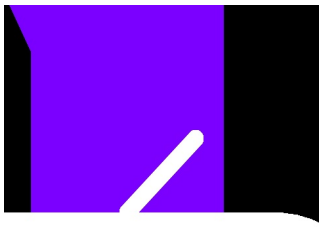
Basic Board Specs	Trace Width/Spacing	8.00/10.00mil+
	Milling Density	91.7080m/m²
	Surface Finish Area	30.94%
	Test Point Count	25

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	Smallest Trace Width	1	Pass 2
	Smallest Trace Spacing	3	Pass 3
	SMD Pad Spacing	2	Pass
	Pad Size	3	Pass
	Hatched Copper Pour	2	Pass
	Annular Ring Size	2	Pass
	Drill to Copper	5	Pass 3
	Signal Integrity	4	Pass
	Copper-to-Board Edge	2	Pass 16
	Holes on SMD Pads	4	Pass
	Open/Shorts (IPC)	1	Fail
PCB Drilling Analysis	Drill Diameter	8	Pass 24
	Drill Hole Density	1	Pass
	Drill Diameter	8	Pass 24
	Drill Spacing	4	Pass
	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
	Missing SMask Opening	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Pass 1 , Fail 2
PCBA Fiducial Analysis	Fiducial Count	1	Pass

ID	Check	Limits	Value	Issue	Image	Position	Qty	Level
1	Silkscreen Spacing _Solder Mask-to-Silkscreen	4,5,6	Error(s) detected	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.		131.78,-106.00	2	Risk