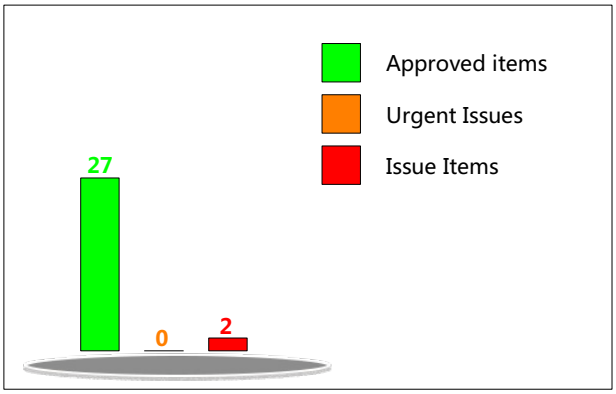
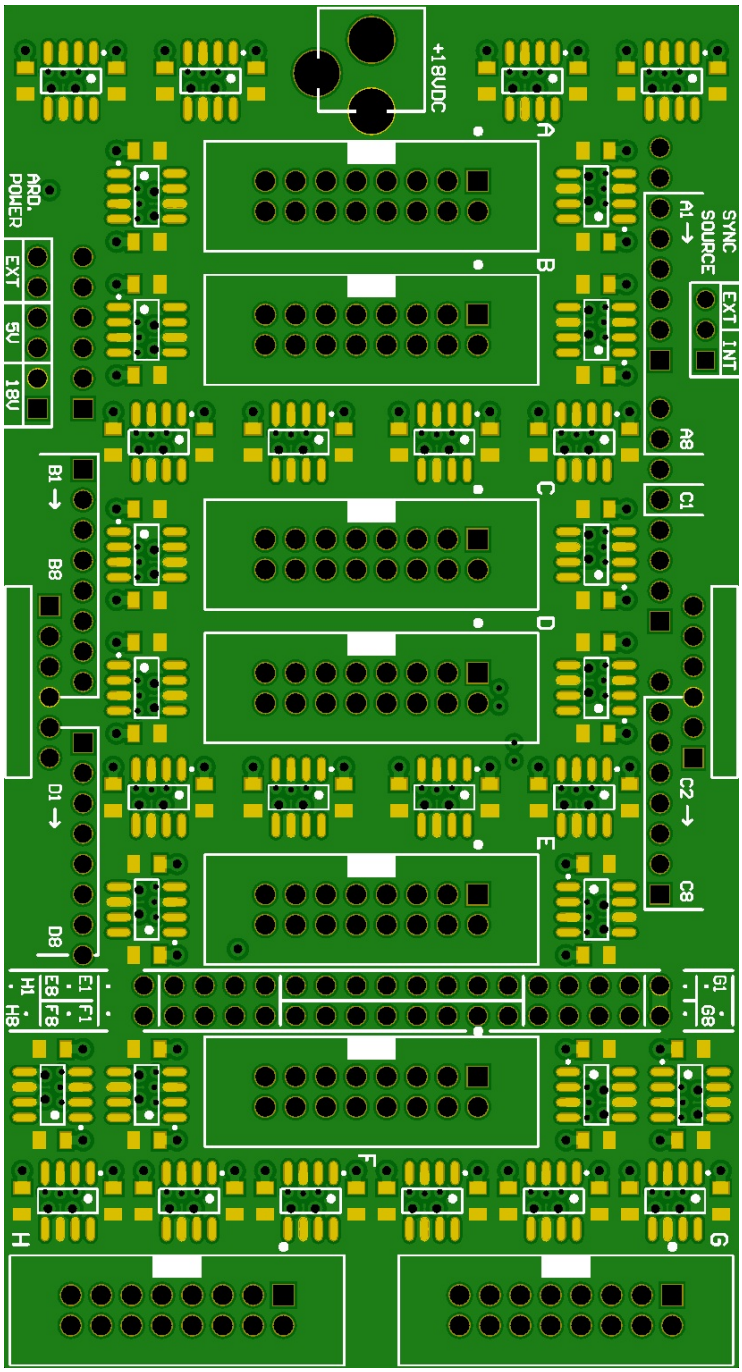
HQDFM Design for Manufacture(DFM) Report

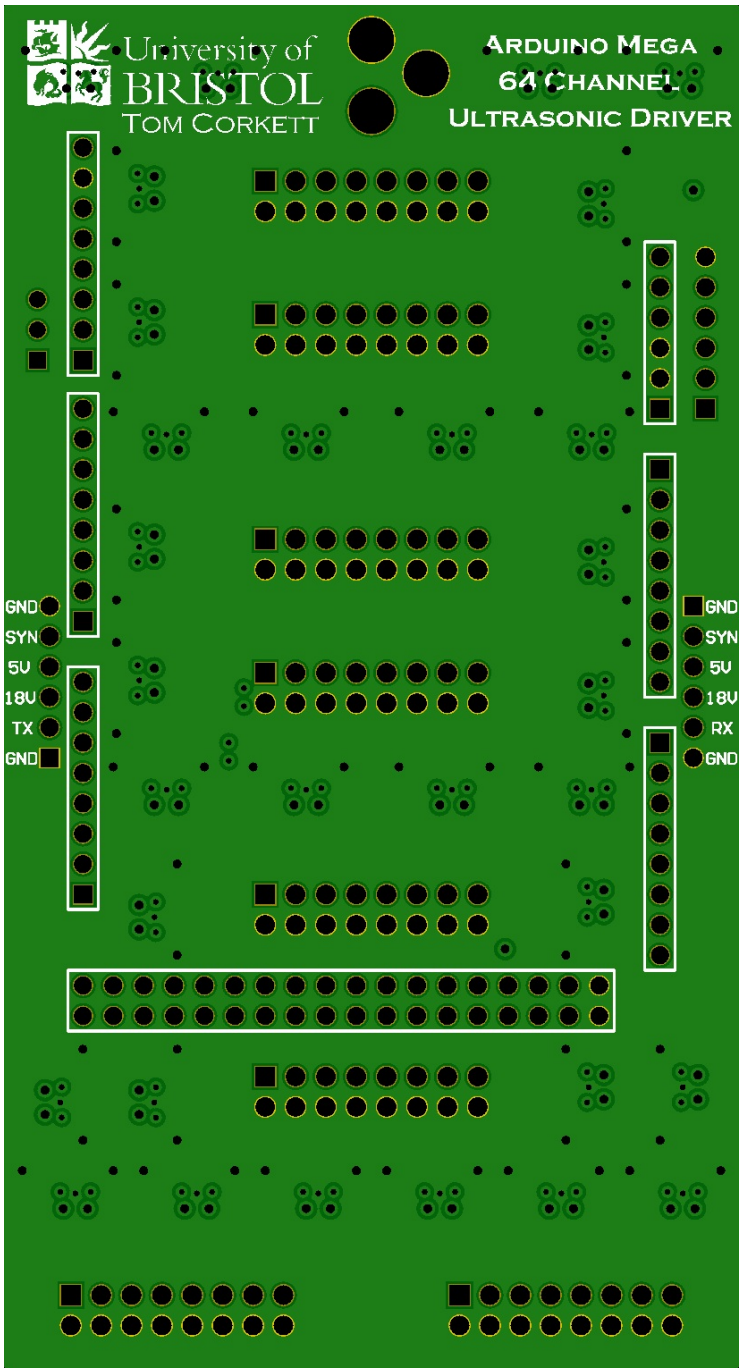
File name: 0000A328399_1

Time: 2025-02-24 Layer count:4 PCB Thickness: 1.60 Quantity: 5 mm

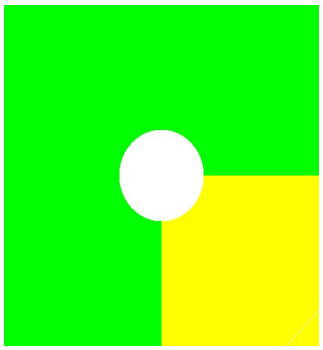


Basic Board Specs	Trace Width/Spacing	10.00/10.00mil
	Milling Density	51.2351m/m²
	Surface Finish Area	33.55%
	Test Point Count	618





Type	Category	No. of Checks	Result
PCB Trace Analysis	Open/Shorts (IPC)	1	Fail
	Signal Integrity	4	Pass
	Smallest Trace Width	1	Pass 3
	Smallest Trace Spacing	3	Pass 131
	SMD Pad Spacing	1	Pass
	Pad Size	3	Pass 4
	Hatched Copper Pour	2	Pass
	Annular Ring Size	2	Pass 4
	Drill to Copper	5	Pass 1023
	Copper-to-Board Edge	2	Pass 1 , Fail 8
	Holes on SMD Pads	4	Pass
PCB Drilling Analysis	Drill Diameter	8	Pass 24
	Drill Hole Density	1	Pass
	Drill Diameter	8	Pass 24
	Drill Spacing	4	Pass 96
	Drill to Board Edge	4	Pass
	Drill Hole Density	1	Pass
	Special Drill Holes	2	Pass
	Drill Hole Errors	3	Pass
PCB Solder Mask Analysis	Solder Mask Dam	2	Pass
	Missing SMask Opening	1	Pass
	Solder Paste Area	1	Pass
PCB Silk Analysis	Silkscreen Spacing	1	Pass
PCBA Fiducial Analysis	Fiducial Count	1	Fail

ID	Check	Limits	Value	Issue	Image	Position	Qty	Level
1	Copper-to-Board Edge_Copper-to-Board Edge	8,15,20	0.00 mm	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. Copper-to-board edge spacing of 0.00mil 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).		0.00,114.05	4	Risk