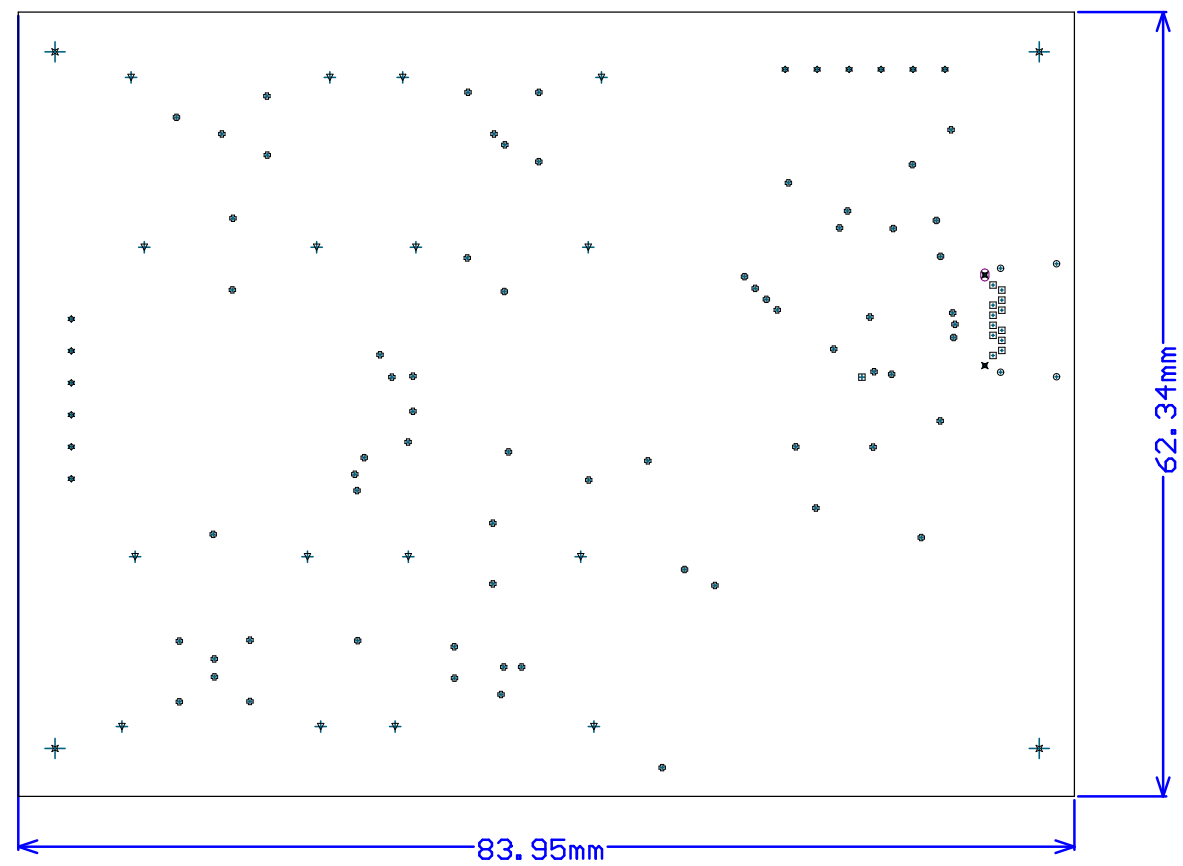




Symbol	Hit Count	Finished Hole Size	Plated	Drill Layer Pair	Via/Pad	Hole Type	Hole Length	Routed Path Length
✕	2	0.650mm (25.59mil)	NPTH	Top Layer - Bottom Layer	Pad	Round	-	-
○	4	0.500mm (19.69mil)	PTH	Top Layer - Bottom Layer	Pad	Slot	1.100mm (43.31mil)	0.600mm (23.62mil)
✕	4	3.200mm (125.98mil)	NPTH	Top Layer - Bottom Layer	Pad	Round	-	-
☆	12	0.900mm (35.43mil)	PTH	Top Layer - Bottom Layer	Pad	Round	-	-
□	13	0.400mm (15.75mil)	PTH	Top Layer - Bottom Layer	(Mixed)	Round	-	-
▽	16	1.800mm (70.87mil)	PTH	Top Layer - Bottom Layer	Pad	Round	-	-
⊕	66	0.300mm (11.81mil)	PTH	Top Layer - Bottom Layer	Via	Round	-	-
	117 Total							

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout

ASSEMBLY NOTES:

1. ALL COMPONENTS SHALL BE RoHS COMPLIANT.
2. ALL UNUSED THROUGH HOLE COMPONENT LOCATIONS SHALL BE FREE OF SOLDER.
3. ALL COMPONENTS SHALL BE MOUNTED FLUSH TO THE BOARD, EXCEPT AS NOTED.
4. FINISHED BOARD SHALL BE FREE OF ALL RESIDUES.
5. ALL LEADS SHALL BE TRIMMED TO A MAXIMUM HEIGHT OF 2mm

THIS PCB TO BE MANUFACTURED TO MEET ALL ACCEPTANCE LEVELS
OF A CLASS 2 PCB PER ANSI/IPC-A-600G.

MATERIAL:	FR-4 or Equivalent		
	☒ MULTILAYER	2 LAYERS	☐ CONTROLLED IMPEDANCE
	Cu WEIGHT EXTERNAL LAYERS	35um FINISHED	
	Cu WEIGHT INTERNAL LAYERS	N/A FINISHED	
	FINISHED OVERALL THICKNESS	1.6 mm	$\pm$ 10 %
	COPPER THIEVING ALLOWED	☐ YES	☒ NO

FINISH:

- ☐ LEAD-FREE HOT AIR LEVELING
- ☒ IMMERSION GOLD
- ☐ IMMERSION TIN
- ☐ SMOBC WITH SELECTIVE GOLD PLATING ON LANDS INDICATED. 1um GOLD OVER 5-10 um NICKEL

SOLDERMASK DYNACHEM EPIC 200 LPI OR EQUIVALENT

SOLDERMASK COLOR	GREEN HIGH GLOSS
------------------	------------------

SILKSCREEN COLOR WHITE

DRILLING: ☐ Blind / Buried Vias

Via Layer Pairs Top-Bottom

ALL HOLES TO BE LOCATED BY THE COORDINATES FROM THE NC DRILL DATA PROVIDED.

USE ARTWORK SET NO. 04-12044

PCB REV []

ALL UL LOGO, MANUFACTURER'S ID, AND DATE CODES SHALL BE PLACED ON THE BOTTOM SIDE UNLESS OTHERWISE INDICATED.

ANY ALTERNATIVES TO THE ABOVE SPECIFICATIONS MUST FIRST BE APPROVED.

Layer	Name	Material	Thickness	Constant	Gerber
	Top Overlay				GTO
	Top Solder	Solder Resist	0.010mm	3.5	GTS
1	Top Layer		0.035mm		GTL
	Dielectric1	FR-4	1.500mm	4.8	
2	Bottom Layer		0.035mm		GBL
	Bottom Solder	Solder Resist	0.010mm	3.5	GBS
	Bottom Overlay				GBO

Total board thickness:	1.590mm
------------------------	---------



TITLE: MCP9604 4 Ch Thermocouple Eval Bd. PART NUMBER: EV191

PCB Layout Contact: Christian Palomares	PLOT NAME / GERBER FILE: 'Fab Drawing (Bottom,Top Layer)'
--	--

Project Owner: Christian Palomares	PCB NUMBER: 04-12044	DOCUMENT NUMBER: 04-12044_FAB-D	DATE: 6/10/2025
PCB FILE NAME: 04-12044.PcbDoc	LAYER NAME: FAB (M4)	NOTES (M1)	REV: 3 10mm