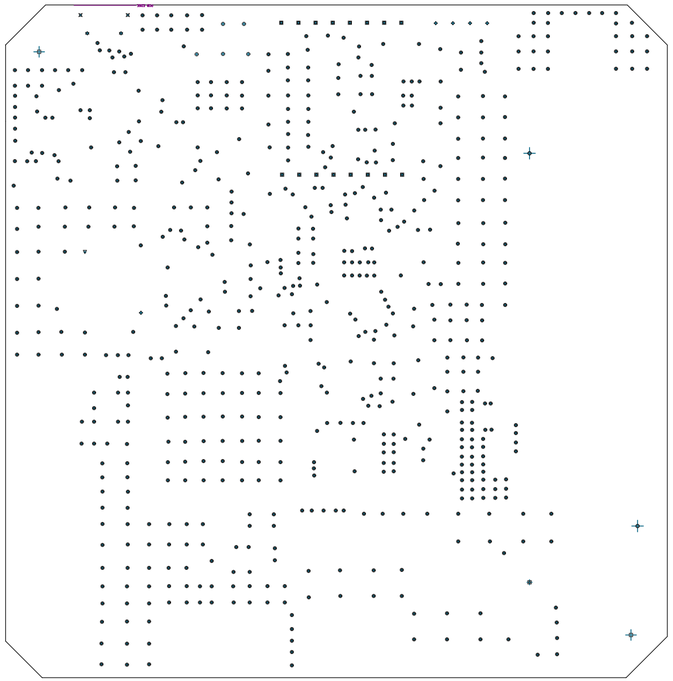


REV	ECO	Comments	Date

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

- [X] MULTILAYER — 2 LAYERS — [ ] CONTROLLED IMPEDANCE
- Cu WEIGHT EXTERNAL LAYERS: 35um FINISHED
- Cu WEIGHT INTERNAL LAYERS: N/A FINISHED
- FINISHED OVERALL THICKNESS: 1.6 mm ± 10 %
- COPPER THIEVING ALLOWED: [ ] YES [X] NO

FINISH:

- [ ] LEAD-FREE HOT AIR LEVELING
- [X] 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: RED

SILKSCREEN COLOR: WHITE

DRILLING: [ ] Blind / Burried Vias

Via Layer Pairs: Top-Bottom

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

USE ARTWORK SET NO. 05-'.ProjectPCBNumber' PCB REV '.ProjectRevisionART'

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

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad Shape	Template	Description	Hole Tolerance (+)	Hole Tolerance (-)	Hole Length	Routed Path Length
▽	1	27.56mil (0.700mm)	PTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c120h70m120p-1				-	-
⊠	1	61.81mil (1.570mm)	NPTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c157hn157				-	-
⌘	2	23.62mil (0.600mm)	PTH	Slot	Top Layer - Bottom Layer	Pad	Rounded Rectangle	s-100h60_130r50				-	-
✡	2	33.47mil (0.850mm)	PTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c145h85				-	-
□	2	125.98mil (3.200mm)	NPTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c0hn320m340				-	-
✜	2	135.83mil (3.450mm)	NPTH	Round	Top Layer - Bottom Layer	Pad	Rounded	c345hn345				-	-
○	5	35.43mil (0.900mm)	PTH	Round	Top Layer - Bottom Layer	Pad	Octagonal	s170h90c50				-	-
◇	5	35.98mil (0.914mm)	PTH	Round	Top Layer - Bottom Layer	Pad	(Mixed)	(Mixed)				-	-
▣	16	36.00mil (0.914mm)	PTH	Round	Top Layer - Bottom Layer	Pad	(Mixed)	(Mixed)				-	-
◎	662	11.81mil (0.300mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v70h30m0mx0				-	-
	698 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

ALL DIMENSIONS ARE DERIVED FROM THE ABOVE LOCATIONS

Layer	Name	Material	Thickness	Constant	Gerber	Board Layer Stack
	Top Overlay		51.18mil (1.300mm)	27.56mil (0.700mm)	GTO	
	Top Solder	Solder Resist	0.010mm	3.5	GTS	
1	Top Layer		0.035mm	-	GTL	
	Dielectric1	FR-4	1.500mm	4.3		
2	Bottom Layer		0.035mm	-	GBL	
	Bottom Solder	Solder Resist	0.010mm	3.5	GBS	
	Bottom Overlay				GBO	
Total board thickness:			1.590mm			

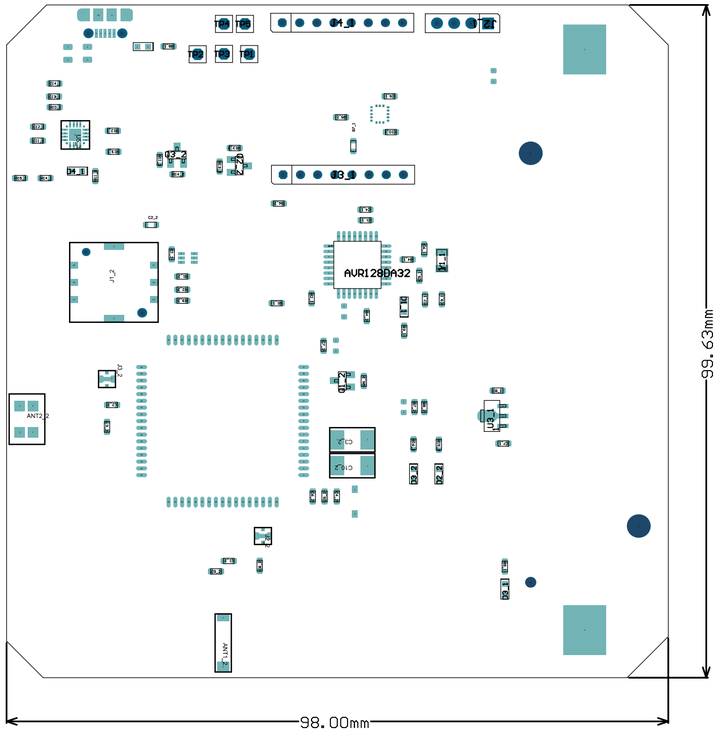
ASSEMBLY NOTES:

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

MICROCHIP
Microchip Technology Inc.
2355 W. Chandler Blvd.
Chandler, AZ 85224

TITLE: '.ProjectTitle'		PART NUMBER: '.ProjectpartNumber'
PCB Layout Contact: '.ProjectDrawnBy'	GERBER FILE: 'Fab Drawing (Bottom,Top Layer)'	
Project Owner: '.ProjectEngineer'	BOARD NUMBER: 04-'.ProjectPCBNumber'	DOCUMENT NUMBER: 04-'.ProjectPCBNumber'-20D2 / DATE: 7/6/2022
PCB FILE NAME: GPS_TRACKER.PcbDoc	LAYER NAME: NOTES (M1)	REV:

REV	ECO	Comments	Date



6. PLACE LABEL ON THE LOCATION INDICATED.
7.

MICROCHIP
Microchip Technology Inc.
2355 W. Chandler Blvd.
Chandler, AZ 85224

TITLE: '.ProjectTitle'

PART NUMBER: '.ProjectpartNumber'

ASSEMBLY VARIANT: Variant of GPS_TRACKER1

PCB Layout Contact: '.ProjectDrawnBy'

GERBER FILE: 'Top Assembly Drawing'

Project Owner: '.ProjectEngineer'

BOARD NUMBER: 04–'.ProjectPCBNumber'

DOCUMENT NUMBER: '.ProjectPCBAssyNumber'–D

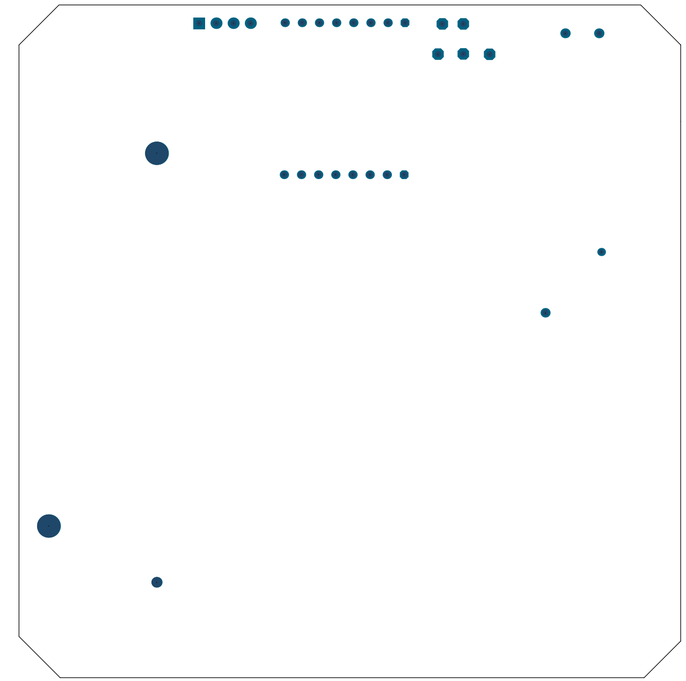
DATE: 7/4/2022

PCB FILE NAME: GPS_TRACKER.PcbDoc

LAYER NAME:

REV:

10mm
400mil



MICROCHIP

Microchip Technology Inc.
2355 W. Chandler Blvd.
Chandler, AZ 85224

TITLE: '.ProjectTitle'			PART NUMBER: '.ProjectpartNumber'
ASSEMBLY VARIANT: Variant of GPS_TRACKER1			
PCB DESIGNER: '.ProjectDrawnBy'	GERBER FILE: 'Bottom Assembly Drawing'		
ENGINEER: '.ProjectEngineer'	BOARD NUMBER: 04–'.ProjectPCBNumber'	DOCUMENT NUMBER: '.ProjectPCBAssyNumber'–D	DATE: 7/4/2022
PCB FILE NAME: GPS_TRACKER.PcbDoc	LAYER NAME: Bottom Assembly	REV: '.ProjectRevisionASSY'	10mm