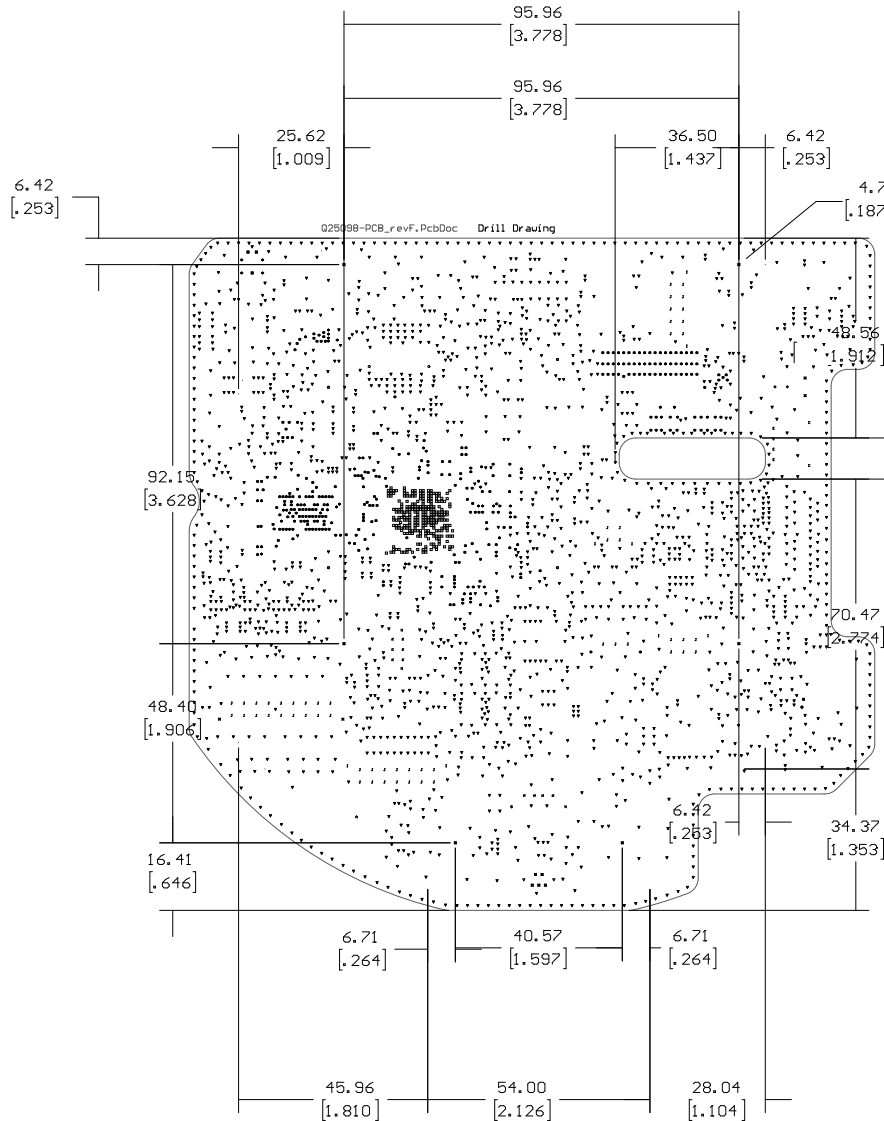


FABRICATION NOTES:

- MATERIAL SELECTION:
370HR OR EQUIVALENT UL RECOGNIZED ZPMV2 MIN. 130C FLAME CLASS V-0 OR BETTER,
MINIMUM CTI RATING OF 175, .062 +/- 0.007 THICK. MATERIAL PER IPC-4101.
SOLDERABLE SURFACES TO BE ENIG (ELECTROLESS NICKEL IMMERSION GOLD) FINISH.
STARTING COPPER WEIGHT INTERNAL 1/2 oz. MINIMUM.
STARTING COPPER WEIGHT EXTERNAL 1/3 oz. MINIMUM.
- SOLDER RESIST: THE USE OF SOLDER RESIST COATING SHALL BE IN
ACCORDANCE WITH THE REQUIREMENTS OF IPC-SM-840. ALL SOLDERABLE
SURFACES ARE TO BE FREE OF SOLDER RESIST. COLOR - GREEN.
USE LIQUID PHOTOMAGEABLE RESIST, MATTE FINISH.
- SILKSCREEN: USE WHITE NON-CONDUCTIVE INK. ALL COMPONENT AND
TESTPOINT LANDS ARE TO BE FREE OF INK.
PLACE UL 94V-0 RATING ON SOLDER SIDE IN SILKSCREEN ONLY.



- MANUFACTURER'S IDENTIFICATION: ADD IN ETCH OR TO SILKSCREEN.
- ELECTRICAL BARE BOARD TEST REQUIRED.
- DRILL SIZES ARE FINISHED SIZES AFTER PLATING.
- FABRICATE TO MEET EU RoHS DIRECTIVE.
- PCB MUST HAVE UL 94V-0 AND CTI RATING MARKED ON ONE SIDE.
- MAX WARP AND TWIST NOT TO EXCEED 0.010 PER LINEAR INCH.
- MIN ANNUAL RING: 0.003. MIN PLATED HOLE WALL THICKNESS 0.001.
- DIMENSIONAL TOL: XX +/- 0.010. XXX +/- 0.005.
- FABRICATE IN ACCORDANCE WITH IPC-600 OR IPC-6012 LATEST REVISION. CLASS 2.
- COPPER THEIVING OF THE SIGNAL LAYERS IS NOT ALLOWED.
- PCB FABRICATOR IS ALLOWED TO ADJUST PARAMETERS TO ACHIEVE REQUIRED TRACE IMPEDANCE +/-10%.
- VIPPO TECHNOLOGY REQUIRED - ALL WAS TO BE NON-CONDUCTIVE EPOXY FILLED AND PLATED OVER.

Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad	Hole Tolerance (+)	Hole Tolerance (-)
□	263	6.00mil (0.152mm)	PTH	Round	Via	3.00mil (0.076mm)	6.00mil (0.152mm)
★	367	8.00mil (0.203mm)	PTH	Round	Via	3.00mil (0.076mm)	8.00mil (0.203mm)
▼	2116	12.00mil (0.305mm)	PTH	Round	Via	3.00mil (0.076mm)	12.00mil (0.305mm)
●	2	12.00mil (0.305mm)	PTH	Round	Via		
○	2	24.00mil (0.610mm)	NPTH	Round	Pad	2.00mil (0.051mm)	2.00mil (0.051mm)
▽	2	26.00mil (0.660mm)	PTH	Slot	Pad	4.00mil (0.102mm)	4.00mil (0.102mm)
⊘	5	28.00mil (0.711mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
E	4	35.00mil (0.889mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
H	4	37.00mil (0.940mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
■	3	39.00mil (0.991mm)	NPTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
⊞	2	40.00mil (1.016mm)	NPTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
I	8	40.00mil (1.016mm)	NPTH	Round	Pad	2.00mil (0.051mm)	2.00mil (0.051mm)
J	2	40.00mil (1.016mm)	NPTH	Slot	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
⊙	55	42.00mil (1.067mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
○	2	42.00mil (1.067mm)	PTH	Round	Pad		
K	6	47.00mil (1.194mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
⊙	2	50.00mil (1.270mm)	NPTH	Round	Pad	2.00mil (0.051mm)	2.00mil (0.051mm)
N	1	90.00mil (2.286mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
G	1	91.00mil (2.311mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
⊙	4	93.00mil (2.362mm)	NPTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
X	2	95.00mil (2.413mm)	PTH	Round	Pad		
C	2	134.00mil (3.404mm)	NPTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
○	4	166.00mil (4.216mm)	PTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
⊞	6	187.00mil (4.750mm)	NPTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
A	1	433.07mil (11.000mm)	NPTH	Round	Pad	3.00mil (0.076mm)	3.00mil (0.076mm)
2866 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

Layer Name	Material	Thickness	Constant	Board Layer Stack
Top Overlay		0.40mil	3.8	
Top Solder	Solder Resist	1.40mil		
1 Top Layer 1	FR-4	4.50mil	4.2	
2 Layer 2 (GND)		0.60mil		
Dielectric 2		4.00mil	4.2	
3 Layer 3 (Sig HS-U)		0.60mil		
Dielectric 3		3.80mil	4.2	
4 Layer 4 (Sig ID)		0.60mil		
Dielectric 4		4.00mil	4.2	
5 Layer 5 (Power +SUS)		0.60mil		
Dielectric 5		8.20mil	4.2	
6 Layer 6 (Power +SUS)		0.60mil		
Dielectric 6		4.00mil	4.2	
7 Layer 7 (Sig U)		0.80mil		
Dielectric 7		3.80mil	4.2	
8 Layer 8 (Sig HS-ID)		0.60mil		
Dielectric 8		4.00mil	4.2	
9 Layer 9 (GND)		0.60mil		
Dielectric 9		4.80mil	4.2	
10 Bottom Layer 10		1.40mil		
Bottom Solder	Solder Resist	0.40mil	3.8	
Bottom Overlay				

Impedance requirements: PCB trace widths and spacings

	100 ohm diff	100 (Trace Spacing)	90 ohm diff	90 (Trace Spacing)	50ohm
Layer 1	5.1	6	7	6	5.1
Layer 2 (Gnd Plane)					
Layer 3	4	6.2	4.7	5	4
Layer 4	4	6.2	4.7	5	4
Layer 5 (Power Plane)					
Layer 6 (Power Plane)					
Layer 7	4	6.2	4.7	5	4
Layer 8	4	6.2	4.7	5	4
Layer 9 (Gnd Plane)					
Layer 10	5.1	6	7	6	5.1