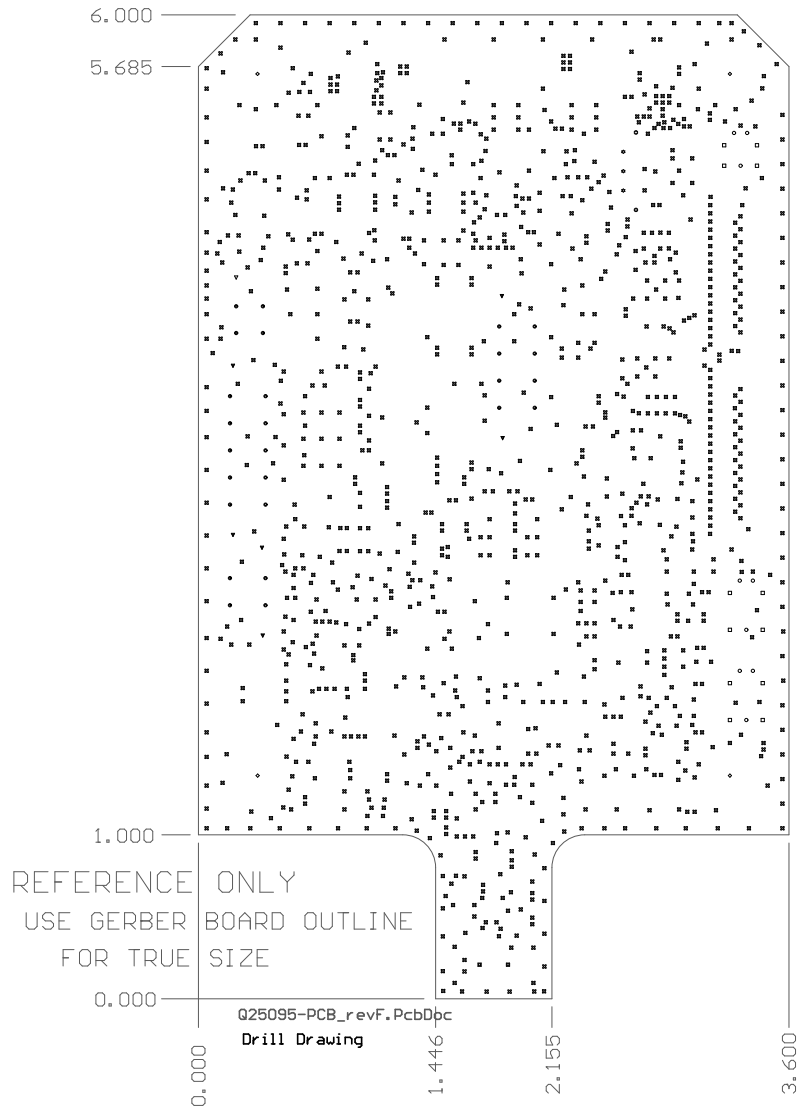


Layer	Name	Material	Thickness	Constant	Board Layer Stack
	Top Overlay				
	Top Solder	Solder Resist	0.60mil	3.8	
1	Top Layer		0.71mil		
	Dielectric 1	PP-014	4.20mil	4.2	
	Dielectric 2	PP-014	4.20mil	4.2	
2	Mid-Layer 1 (GND)		1.20mil		
	Dielectric 3	Core-042	39.00mil	4.6	
3	Mid-Layer 2 (POWER)		1.20mil		
	Dielectric 4	PP-014	4.20mil	4.2	
	Dielectric 5	PP-014	4.20mil	4.2	
4	Bottom Layer		0.71mil		
	Bottom Solder	Solder Resist	0.60mil	3.8	
	Bottom Overlay				

4 LAYER STACK-UP:

TOP PASTE MASK: Q25095-PCB_revF.GTP
SILKSCREEN TOP SIDE: Q25095-PCB_revF.GTO
SOLDERMASK TOP SIDE: Q25095-PCB_revF.GTS
LAYER 1 CIRCUIT TOP SIDE: Q25095-PCB_revF.GTL
LAYER 2 GND PLANE: Q25095-PCB_revF.G1
LAYER 3 POWER: Q25095-PCB_revF.G2
LAYER 4 CIRCUIT BOTTOM SIDE: Q25095-PCB_revF.GBL
SOLDER MASK BOTTOM SIDE: Q25095-PCB_revF.GBS
SILKSCREEN BOTTOM SIDE: Q25095-PCB_revF.GBO

NC DRILL: Q25095-PCB_revF.TXT
Q25095-PCB_revF.GD1 DRILL DRAWING
Q25095-PCB_revF.GM7 FABRICATION DRAWING



PCB FABRICATION NOTES:

1. 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 oz. MINIMUM
STARTING COPPER WEIGHT EXTERNAL 1/2 oz. MINIMUM
2. 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.
3. 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.
4. MANUFACTURER'S IDENTIFICATION: ADD IN ETCH OR TO SILKSCREEN.
5. ELECTRICAL BARE BOARD TEST REQUIRED.
6. DRILL SIZES ARE FINISHED SIZES AFTER PLATING.
7. FABRICATE TO MEET EU RoHS DIRECTIVE.
8. PCB MUST HAVE UL 94V-0 AND CTI RATING MARKED ON ONE SIDE.
9. MAX WARP AND TWIST NOT TO EXCEED 0.010 PER LINEAR INCH.
10. MIN ANNULAR RING: 0.003. MIN PLATED HOLE WALL THICKNESS 0.001.
11. DIMENSIONAL TOL: XX +/- 0.010. XXX +/- 0.005.
12. FABRICATE IN ACCORDANCE WITH IPC-600 OR IPC-6012 LATEST REVISION. CLASS 2.
13. COPPER THIEVING OF THE SIGNAL LAYERS IS ALLOWED. SPACING TO ANY EXISTING BOARD FEATURE TO BE 0.060 MINIMUM.
14. 0.0102" EXTERNAL TRACES REQUIRE 90 OHM DIFFERENTIAL IMPEDANCE. PCB FABRICATOR IS ALLOWED TO ADJUST PARAMETERS TO ACHIEVE REQUIRED TRACE IMPEDANCE +/-10%.
15. VIPO TECHNOLOGY REQUIRED - ALL VIAS TO BE TYPE VII NON-CONDUCTIVE EPOXY FILLED AND PLATED OVER.

Symbol	Count	Hole Size	Via/Pad	Plated	Hole Type	Pad Shape	Hole Tolerance
⊗	1231	12.00mil (0.3050mm)	(Mixed)	PTH	Round	Rounded	+3.00mil/-12.00mil
■	2	35.00mil (0.8890mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
○	9	39.00mil (0.9910mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
☆	3	42.00mil (1.0670mm)	Pad	PTH	Round	(Mixed)	+/-3.00mil
⊙	2	50.00mil (1.2700mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
⦿	26	55.00mil (1.3970mm)	Pad	PTH	Round	(Mixed)	+/-3.00mil
□	12	93.00mil (2.3620mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
▼	6	118.00mil (2.9970mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
▽	1	125.00mil (3.1750mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
◇	4	221.00mil (5.6130mm)	Pad	NPTH	Round	Rounded	+/-2.00mil
1296 Total							