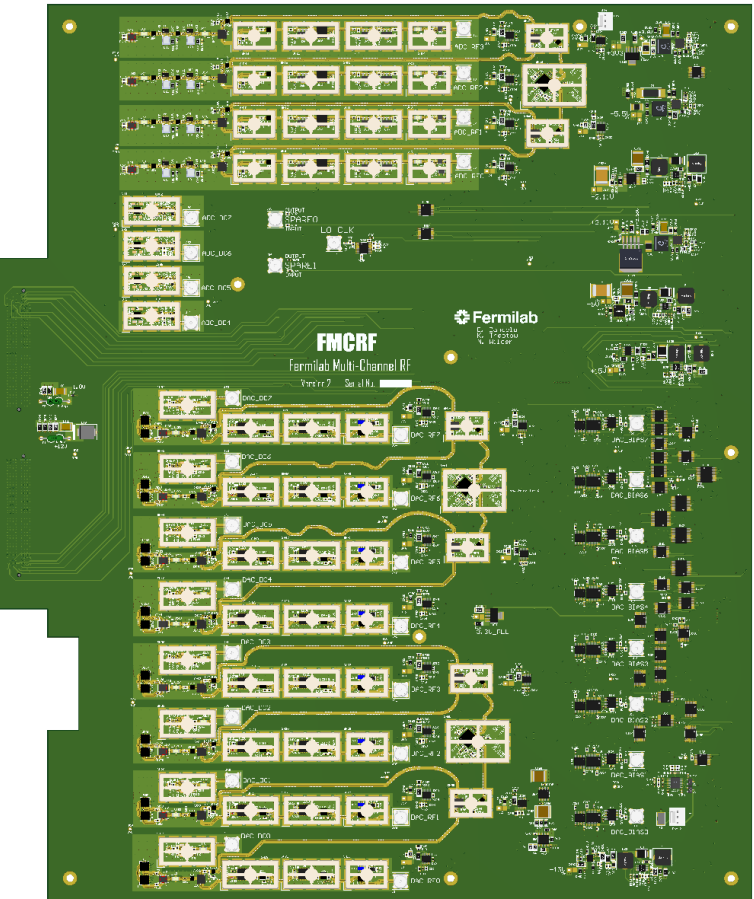


Transmission Line Structure Table

Impedance Id	Transmission Line	Target Impedance	Calculated Impedance	Trace layer	Lower Trace Width	Upper Trace Width	Reference layers	Substack
1	Coated Microstrip	50	56.27	Top Layer	16.00mil	16.00mil	GND1	Board Layer Stackup
2	Offset Stripline	50	49.05	Signal 1	6.00mil	6.00mil	GND1,GND2	Board Layer Stackup
3	Offset Stripline	50	49.10	Signal 2	6.00mil	6.00mil	GND2,GND3	Board Layer Stackup
4	Offset Stripline	50	49.10	Signal 3	6.00mil	6.00mil	GND3,Signal 4	Board Layer Stackup
5	Offset Stripline	50	49.10	Signal 4	6.00mil	6.00mil	Signal 3,GND4	Board Layer Stackup
6	Offset Stripline	50	49.10	Signal 5	6.00mil	6.00mil	GND4,GND5	Board Layer Stackup
7	Offset Stripline	50	49.05	Signal 6	6.00mil	6.00mil	GND5,GND6	Board Layer Stackup
8	Coated Microstrip	50	56.27	Bottom Layer	16.00mil	16.00mil	GND6	Board Layer Stackup

Realistic View



Drill Table

Symbol	Count	Hole Size	Plated	Hole Tolerance
□	23499	8.00mil	Plated	
▼	181	12.00mil	Plated	
▣	74	15.00mil	Plated	
◇	215	20.00mil	Plated	
◎	1	25.00mil	Plated	
✱	2	35.00mil	Plated	
▽	6	38.00mil	Plated	
○	53	40.00mil	Plated	
⊠	4	54.00mil	Plated	
⊞	4	57.09mil	Non-Plated	
✱	140	67.00mil	Plated	
☆	10	125.00mil	Plated	
	24189 Total			

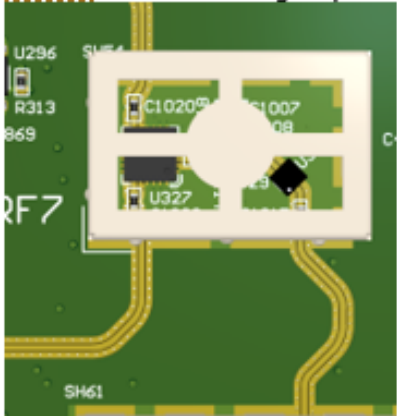
RF Gen 4

Layer Stack Legend

Material	Layer	Thickness	Dielectric Material	Type	Gerber
	Top Overlay			Legend	GTO
Surface Material	Top Solder	0.40mil	Solder Resist	Solder Mask	GTS
Copper	Top Layer	0.70mil		Signal	GTL
		9.00mil	R4350B	Dielectric	
Copper	GND1	0.70mil		Internal Plane	GP1
		6.00mil	R4350B	Dielectric	
CF-003	Signal 1	0.70mil		Signal	G1
Prepreg		6.00mil	R4350B	Dielectric	
Copper	GND2	0.70mil		Internal Plane	GP2
Prepreg		6.00mil	R4350B	Dielectric	
CF-004	Signal 2	0.69mil		Signal	G2
Prepreg		6.00mil	R4350B	Dielectric	
CF-004	GND3	0.69mil		Internal Plane	GP3
Prepreg		6.00mil	R4350B	Dielectric	
CF-004	Signal 3	0.69mil		Signal	G3
		6.00mil	R4350B	Dielectric	
CF-004	Signal 4	0.69mil		Signal	G4
Prepreg		6.00mil	R4350B	Dielectric	
CF-004	GND4	0.69mil		Internal Plane	GP4
Prepreg		6.00mil	R4350B	Dielectric	
CF-004	Signal 5	0.69mil		Signal	G5
Prepreg		6.00mil	R4350B	Dielectric	
Copper	GND5	0.70mil		Internal Plane	GP5
Prepreg		6.00mil	R4350B	Dielectric	
CF-003	Signal 6	0.70mil		Signal	G6
		6.00mil	R4350B	Dielectric	
Copper	GND6	0.70mil		Internal Plane	GP6
		9.00mil	R4350B	Dielectric	
Copper	Bottom Layer	0.70mil		Signal	GBL
Surface Material	Bottom Solder	0.40mil	Solder Resist	Solder Mask	GBS
	Bottom Overlay			Legend	GBO
Total thickness: 94.53mil					

ReadMe File for RF Gen4

- The project/board name is "RF Gen4"
- Quantity of boards is 13
- Turnaround time for first board is two weeks. If this is not possible, let me know what is.
- This quote should take into consideration the following:
 - This project is for manufacture of pc boards, purchase of components, and assembly.
 - Assembly will be done in 2 stages. The first stage will result in 1 pc board fully assembled.
 - The second stage will have the remaining boards assembled once we approve the design.
- Class: Fabricate PCB in accordance with IPC-6012A, Class 2; per IPC-6011 using customer data.
- Rout Type: ARRAY-ROUT & RETAIN
- The board is 14 layers. Stackup is defined in the included drawing RF Gen4 Spec Sheet.pdf
- 5mil line, 5 mil space
- The final thickness needs to be 0.095" or less. If the layer thicknesses in "RF Gen4 Spec Sheet.pdf" need to be modified to achieve the required impedance control, that is permitted as long as board thickness doesn't exceed 0.095".
- If copper thieving is needed to achieve copper balance and to prevent warping, you are permitted to do so.
- The board shall be manufactured using Rogers 4350B. If you wish to use an equivalent, please contact me so we can discuss it.
- There are some vias that need to be filled with non-conductive epoxy, overplated and capped. They are 18 mill pads with 8 mill drills, and any 12 mil drill holes.
- This pc board has through hole and blind vias. The blind vias exist in the top layer to the adjacent GND plane layer.
- Vias shall be tented on top and bottom layer
- Copper weight is ½ oz.
- Material finish is ENIG
- SMOBC (Solder mask over bare copper)
- Hole sizes are finished after plating.
- Testing: Netlist testing for connectivity, opens, and shorts
- Copper plating on PTH: minimum 1 mil plating in the hole
- Drill hole tolerance: <50mil +/- 3mil
- Drill hole tolerance: >50mil +/- 7mil
- This board is impedance controlled.** All signals are 50 ohms to the adjacent GND plane. The MOST IMPORTANT signals are the 16 mil wide co-planar waveguide signals on the top layer. Those signals have GND vias stitched parallel to the traces. Soldermask has been removed from these traces as shown in the picture below. This must be taken into account when calculating co-planar waveguide impedance.



- 100 ohm Differential to adjacent GND layers for the 5 mil traces with 10 mil spacing on Signal layer 1 and Signal layer 6.
- Any questions or suggestions for modifications must be discussed with me.