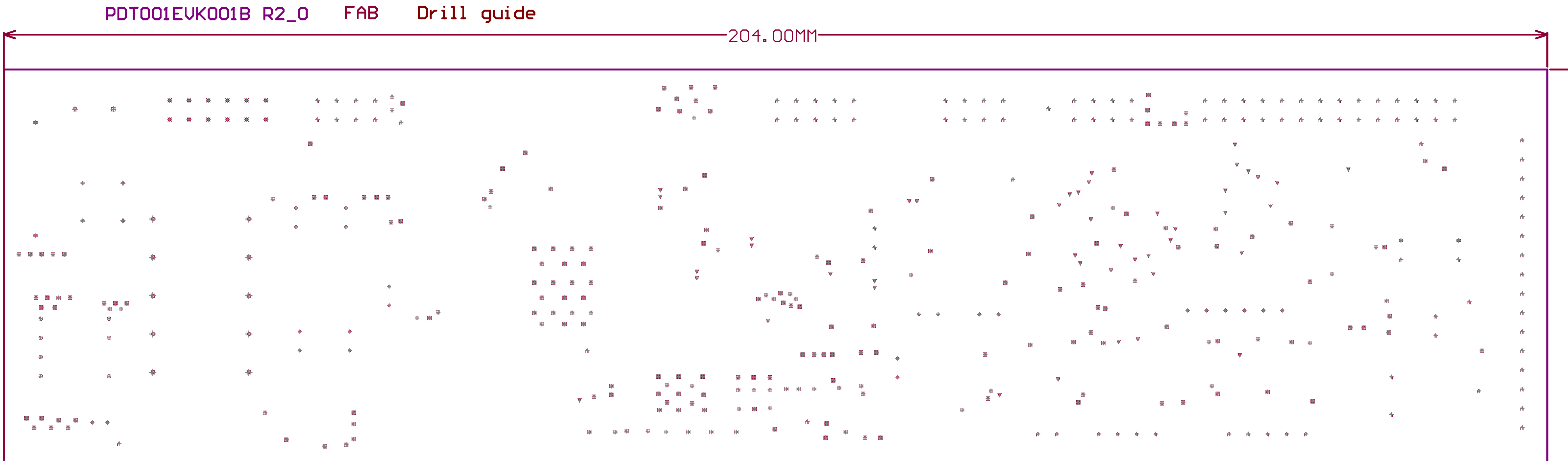


Layer	Layer	Material	Thickness	Constant
1	Top SolderMask	Solder Resist	0.002 mm	DK 4.2
2	Top Copper	Copper	0.035 mm	
3	PrePreg	2116	0.128 mm	DK 4.0
4	PrePreg	2113	0.088 mm	DK 4.0
5	Inner 1	Copper	0.035 mm	
6	Core	FR4	1.0 mm	
7	Inner 2	Copper	0.035 mm	
8	PrePreg	2113	0.088 mm	DK 4.0
9	PrePreg	2116	0.128 mm	DK 4.0
10	Bottom Copper	Copper	0.035 mm	
11	Bottom SolderMask	Solder Resist	0.002 mm	DK 4.2
			1.612 mm	

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad
⊞	2	47.24mil <1.200mm	PTH	Round	Top Layer - Bottom Layer	Pad
⊞	2	51.18mil <1.300mm	PTH	Round	Top Layer - Bottom Layer	Pad
☆	6	39.37mil <1.000mm	PTH	Round	Top Layer - Bottom Layer	Pad
○	7	36.00mil <0.914mm	PTH	Round	Top Layer - Bottom Layer	Pad
⊞	10	66.93mil <1.700mm	PTH	Round	Top Layer - Bottom Layer	Pad
✕	12	43.31mil <1.100mm	PTH	Round	Top Layer - Bottom Layer	Pad
◇	24	31.50mil <0.800mm	PTH	Round	Top Layer - Bottom Layer	Pad
▽	44	12.00mil <0.305mm	PTH	Round	Top Layer - Bottom Layer	Via
⋈	106	39.37mil <1.000mm	PTH	Round	Top Layer - Bottom Layer	Pad
⊞	217	20.00mil <0.508mm	PTH	Round	Top Layer - Bottom Layer	Via
	430 Total					



NOTE:

1. PCB Laminate material shall be FR4 Grade.
2. PCB Size 240X 52 MM, Four Layers PCB And Board Thicknes is 1.6MM
3. Top and Bottom Layer Finishing copper thickness is 35micron.
4. All exposed conductive pattern areas not covered with solder mask.
5. Board Finising Hot Air Solder Level(HAL).
6. Apply liquid photo imageable solder mask (color Green) per IPC-SM-840, class H, on both sides of the board over bare copper.
7. Silkscreen shall be white, permanent, organic, non-conductive ink.
There shall be no silkscreen on any solderable component pad.
8. PCB Shall be 100% netlist electrical verification.