



Layer	Layer	Material	Thickness	Constant
1	Top SolderMask	Solder Resist	0.002 mm	DK 4.2
2	Top Copper	Copper	0.070 mm	
3	Core	FR4	1.5 mm	
4	Bottom Copper	Copper	0.070 mm	
5	Bottom SolderMask	Solder Resist	0.002 mm	DK 4.2
			1.680 mm	

CUTOUT: Dimension

Ⓐ 105mm × 64mm

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Hole Length	Routed Path Length
K	2	1.400mm (55.12mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
B	2	1.600mm (62.99mil)	PTH	Slot	Top Layer - Bottom Layer	Pad	9.200mm (362.21mil)	7.600mm (299.21mil)
▣	2	2.500mm (98.43mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
M	2	3.000mm (118.11mil)	PTH	Slot	Top Layer - Bottom Layer	Pad	8.000mm (314.96mil)	5.000mm (196.85mil)
E	6	2.100mm (82.68mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
◇	8	3.200mm (125.98mil)	NPTH	Round	Top Layer - Bottom Layer	Pad	-	-
⊕	8	12.000mm (472.44mil)	NPTH	Round	Top Layer - Bottom Layer	Pad	-	-
⊙	10	0.700mm (27.56mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
⊗	20	0.406mm (16.00mil)	PTH	Round	Top Layer - Bottom Layer	Via	-	-
P	20	0.800mm (31.50mil)	NPTH	Round	Top Layer - Bottom Layer	Pad	-	-
F	22	1.200mm (47.24mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
O	24	1.000mm (39.37mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
D	24	1.016mm (40.00mil)	NPTH	Slot	Top Layer - Bottom Layer	Pad	5.080mm (200.00mil)	4.064mm (160.00mil)
✕	24	1.500mm (59.06mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
□	64	1.000mm (39.37mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
N	90	1.800mm (70.87mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
★	92	1.100mm (43.31mil)	PTH	Round	Top Layer - Bottom Layer	Pad	-	-
C	844	0.508mm (20.00mil)	PTH	Round	Top Layer - Bottom Layer	Via	-	-
1264 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

NOTE:

- PCB Laminate material shall be FR4 Grade.
- PCB Size (285x140mm & 30x109mm), Two Layers PCB And Board Thicknes is 1.6MM
- Top and Bottom Layer Finishing copper thickness is 100micron.
- All exposed conductive pattern areas not covered with solder mask.
- Board Finising Hot Air Solder Level(HAL).
- Apply liquid photo imageable solder mask (color Green) per IPC-SM-840, class H, on both sides of the board over bare copper.
- Silkscreen shall be white, permanent, organic, non-conductive ink. There shall be no silkscreen on any solderable component pad.
- PCB Shall be 100% netlist electrical verification.