



Common PCB Layout Software Gerber File Extension Comparison

	Altium / Protel	Eagle	KiCad	Orcad	PADS
Native Project Save File	.PCBPRJ	.BRD			.PCB
Top Silk	.GTO	.PLC / .TSK	F.SILKS	.SST	ST.PHO
Top Mask	.GTS	.STC / .TSM	F.MASK	.SMT	MT.PHO
Top Paste	.GTP	.CRC / .TSP	F.PASTE	.SPT	PT.PHO
Top Copper	.GTL	.CMP / .TOP	F.CU	.TOP	L1.PHO
Inner Signal	.G1, .G2, etc	.LY2, .LY3, etc	--	.IN1, .IN2, etc	L2.PHO, L3.PHO, etc
Inner Plane	.GP1, .GP2, etc	--	--	--	--
Bottom Copper	.GBL	.SOL / .BOT	B.CU	.BOT	L4.PHO (L2 if only 2 layers)
Bottom Paste	.GPB	.CRS / .BSP	B.PASTE	.SPB	PB.PHO
Bottom Mask	.GBS	.STS / .BSM	B.MASK	.SMB	MB.PHO
Bottom Silk	.GBO	.PLS / .BSK	B.SILK	.SSB	SB.PHO
Drill Drawing	.GD1	.DIM / .MIL / .GML	--	--	D1.DWG
Drill Guide	.GG1	.DIM / .MIL / .GML	--	--	--
Keep Out	.GKO	--	--	--	--
Board Outline	.GM1 / .GM3	.DIM / .MIL / .GML	EDGE.CUTS	.FAB	--
CNC Drill	.DRL / .TXT	.DRL / .TXT	--	.TAP	ANC.DRL
Excellon Tool List	--	--	--	--	--
Excellon Drill Description	--	.DRD	--	--	--
Notes	.GKO layer is sometimes used incorrectly as a board outline.	Extensions will change depending on software version.	These are internal file names to KiCad. Protel naming can be selected when exporting Gerber.		

Blanks indicate there isn't a specific file for this layer or none was found at the time the chart was compiled. If you have additions or corrections, please contact www.pcbprime.com