File extensions used to identify each gerber file

When you generate the Gerber output a series of files are created, each one corresponding to one of the layers enabled in the Gerber setup. These files are then loaded into a Gerber photoplotter, which produces the necessary phototools for PCB manufacture.
Each Gerber file is given the name of the PCB document, with a unique extension that identifies that layer and plot type. For example, the Top layer Gerber file for a PCB called MyDesign will be saved as MyDeisgn.GTL, to indicate "Gerber Top Layer". Because each design normally generates numerous Gerber files, these extensions help identify each file.

We recommend that you follow this convention which conforms to general industry practice. The following table shows the extensions that are used:

Top Overlay	.GTO
Bottom Overlay	.GBO
Top Layer	.GTL
Bottom Layer	.GBL
Mid Layer 1, etc.	.G1, .G2, etc
Power Plane 1, etc.	.GP1, GP2, etc
Mechanical Layer 1, etc.	.GM1, .GM2, etc
Top Solder Mask	.GTS
Bottom Solder Mask	.GBS
Top Paste Mask	.GTP
Bottom Paste Mask	.GBP
Drill Drawing	.GDD
Drill Drawing; Top to Mid 1, Mid2 to Mid 3, etc.	.GD1, GD2, GD3, etc.
Drill Guide	.GDG
Drill Guide; Top to Mid 1, Mid 2 to Mid 3, etc	.GG1, GG2, GG3, etc.
Pad Master, Top	.GPT
Pad Master, Bottom	.GPB
Keep Out Layer	.GKO
Gerber Panels	.P01, .P02, etc.