Top overlay	.GTO	Ground Plane	.GGD
Top layer	.GTL	Power Plane	.GPW
Mid layer 1	.GM1	Top Solder Mask	.GTS
Mid layer 2	.GM2	Bottom Solder Mask	.GBS
Mid layer 3	.GM3	Top Paste Mask	.GTP
Mid layer 4	.GM4	Bottom Paste Mask	.GBP
Bottom overlay	.GBO	Drill Drawing	.GDD
Bottom layer	.GBL	Drill Guide	.GDG
Keep Out layer	.GKO	Pad Master	.GPM

Above table is the Gerber Extension names in the sx and nx-Amp PCB files.

To make or order PCB's you will need the layers that are arrowed in RED. Arrows in GREEN are optional if you want solder mask and silk screen

Some PCB houses use the pad master, others just use CAM software to pull it from the top and bottom layer files.

You will also need to include the .DRL files. This is the file that tells the drilling machine what size holes to drill in the PCB fabrication process. The DRL file takes the format "filename.DRL".

Finally, the "filename.APT file should also be included for PCB orders.

Here are some links to Free Gerber Viewers

http://www.easylogix.de/products_detail.php?prog_id=1

http://circuitpeople.com/ - you ypload the file to view it

http://www.pcb-pool.com/ppus/service_downloads.html http://www.pcb-pool.com/ppus/service_downloads.html

http://www.numericalinnovations.com/ind ... stom&ID=37

http://www.bot-thoughts.com/2011/09/free-windows-gerber-viewers.html