Notes about the proper Gerber file preparation

To eliminate errors and shorten preparation time for PCB manufacturing, please consider the following

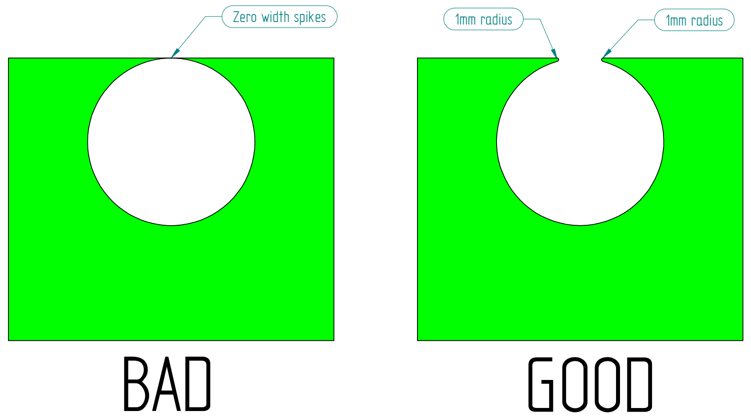
<u>GerberX file naming</u>

Please name the Gerber layers accordingly. Lets say that PCB name is GPCB. Then name the layers following

Layer	Name	Comment
Top Layer	GPCB_Top.gbr	Top Copper layer
Bottom Layer	GPCB_Bottom.gbr	Bottom Copper layer
Inner Layer 1	GPCB_Inner1.gbr	1.st layer from Top
Inner Layer 2	GPCB_Inner2.gbr	2nd layer..
Top Soldermask	GPCB_TopSolder.gbr	Top solder mask (pads without soldermask)
Bottom Soldermask	GPCB_BotSolder.gbr	Bottom solder mask (pads without soldermask)
Board Outline	GPCB_Outline.gbr	We treat outlines and slots as zero-width
Plated Holes	GPCB_plated.txt	Holes drilled before copper plating
Unplated Holes	GPCB_unplated.txt	Holes drilled after soldermask
Plated Slots	GPCB_SlotPlated.gbr	Slots milled before plating
Unplated Slots	GPCB_SlotUnplated.gbr	Slots milled after soldermask
Top Paste	GPCB_TopPaste.gbr	For topside stencil making
Bottom Paste	GPCB_BotPaste.gbr	For bottomside stencil
Top Silkscreen	GPCB_TopSilk.gbr	Top silkscreen (marking)
Bottom Silkscreen	GPCB_BotSilk.gbr	Bottom silkscreen (marking)
Mechanical comment	GPCB_Mech.gbr	Free layer with drawings and comments. Used only for information and description

- All layers must be „viewed from top side“. Dont use mirror !
- Do not use „predrilling“ for large holes
- Unplated holes should be at least 0.2mm away from copper, unless neccesary for connections
- For plated holes, specify final size. Dont make any plating correction (this is our CAM job)
- Generally, use 0.24 / 0.24 track/clearance minimum
- Do not make outlines for holes.
- Silkscreen marking should have at least 0.2mm width, otherwise its unstable and ugly
- With files package, please include only relevant files.
- Dont make overlapping holes which are in both unplated/plated files.
- Please leave at least 0.3mm free space between border and copper (unless its neccesary for edge contacts)
- For stencils, please specify if you want us to undersize pads. If the stencil pad is the same size as soldermask pad, then we usually undersize by 10%. For difficult components (especially power semiconductors with underside cooling pads !) always use exact stencil cutout from component datasheet. If in doubt, consult us. Remember that wrong stencil is responsible for 80% manufacturing problems.

- Do not make cutouts which are tangential with outline. If needed, please trim and chamfer the edges properly. Ask for more information if needed ! Check the drawing



- For fuse or spark-gap design on PCB consult us. We have seen lots of errors from our customers doing it, which makes these features usually pointless.
- For automatic pick&place line (mass production) please consider following
  - Panel corners should be rounded (3mm is good). This helps with conveyor movement
  - Design board breakout so no sanding or mechanical working is needed (consult us).
  - For large panels, use local fiducials (consult us if needed)