

Name	Kicad file name or layer name	ALPWAN file name
Top Assembly	F.Fab	TAY-XXXXX-XXX.pdf
Top Solder Paste	xxxx-F_Paste.gbr for the component-side solder paste	TSP-XXXXX-XXX.gbr
Top Placement File	xxxx-top.pos	TPF-XXXXX-XXX.txt
Placement File	xxxx-all.pos	PFE-XXXXX-XXX.txt
Bottom Placement File	xxxx.bottom.pos	BPF-XXXXX-XXX.txt
Bottom Solder Paste	xxxx-B_Paste.gbr for the copper-side solder paste	BSP-XXXXX-XXX.gbr
Bottom Assembly	B.Fab	BAY-XXXXX-XXX.pdf
Schematic	n/a	SCH-XXXXX-XXX.pdf
Bill Of Materials	xxxx.csv (convert to Excel)	BOM-XXXXX-XXX.xlsx
Bill Of Materials	Also available in PDF	BOM-XXXXX-XXX.pdf
Board Mechanical Drawing	Dwgs.User	BMD-XXXXX-XXX.pdf
Top 3D View	n/a	T3D-XXXXX-XXX.png
3D View	n/a	3DV-XXXXX-XXX.png
Bottom 3D View	n/a	B3D-XXXXX-XXX.png

GERBER file format: The format used by KiCad Pcbnew is RS274X 4.6, Metric, Leading zero omitted, Abs