

## Gerber files explanation

We exported Gerber files from Eagle .brd file using build in CAM processor settings (gerb274x-mill-drill.cam). Gerber files were inspected with “Online Gerber-Viewer” tool (<http://www.gerber-viewer.com/>)

Following files are included:

File	Description	Output Device	Contains Eagle layers. Name (number)
.cmp	Copper components side	GERBER_RS274X_26	Top (1), Pad (17), Vias (18)
.sol	Copper solder side	GERBER_RS274X_26	Bottom (16), Pad (17), Vias (18)
.stc	Solder stop mask component side	GERBER_RS274X_26	tStop (29)
.sts	Solder stop mask solder side	GERBER_RS274X_26	bStop (29)
.dpv	Drills for pads and vias	EXCELLON_24	Drills (44)