DFM analysis report_JLCDFM

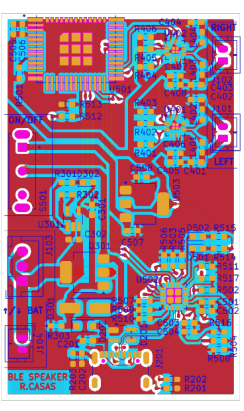
File name: **Gerber.zip**

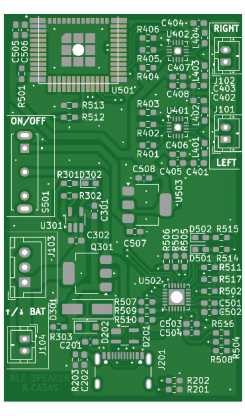
Report generated at: **2025-11-17 09:20:20**

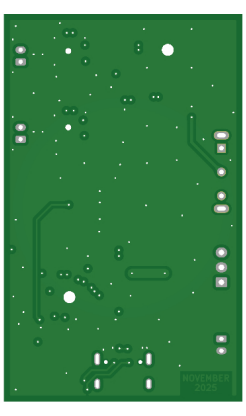
PCB layers: **4**

PCB size: **4x6.5cm**

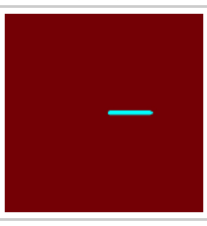
Analyze project: ☑ **PCB DFM** ☐ **SMT DFM**



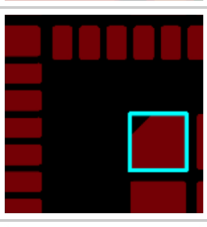




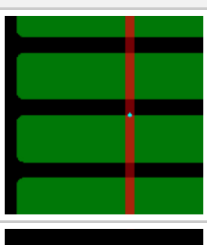
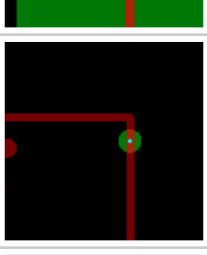
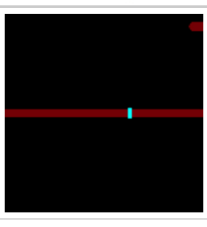
PCB DFM>Routing layer analysis

Analyze project	Analysis results	PCB screenshot	Layer distribution	Statistics
Sharp trace corner (Check for sharp corners in traces)	null Warning		Gerber\PCB_BLE_Speaker-F_Cu.gbr Gerber\PCB_BLE_Speaker-B_Cu.gbr	Danger: 0 Warning: 11 Good: 0
BGA pad (Check BGA pads on the board)	No	No	No	Danger: 0 Warning: 0 Good: 0
Via placed within a pad (Check if there is a via placed within a pad)	No	No	No	Danger: 0 Warning: 0 Good: 0
Trace to board edge (Detect traces too close to the board edge)	No	No	No	Danger: 0 Warning: 0 Good: 0
Trace spacing (Measure spacing between adjacent parallel traces)	0.2mm Good		Gerber\PCB_BLE_Speaker-F_Cu.gbr	Danger: 0 Warning: 0 Good: 12
Unconnected trace end (Free-standing trace ends not connected to pads)	null Warning		Gerber\PCB_BLE_Speaker-F_Cu.gbr Gerber\PCB_BLE_Speaker-B_Cu.gbr	Danger: 0 Warning: 43 Good: 0
Trace width (Trace width information)	0.13mm Good		Gerber\PCB_BLE_Speaker-F_Cu.gbr	Danger: 0 Warning: 0 Good: 21
Fiducial (Detect fiducial marks on the board)	No	No	No	Danger: 0 Warning: 0 Good: 0
Pad to board edge (Measure distance of pads from the board edge)	No	No	No	Danger: 0 Warning: 0 Good: 0
Pad spacing (Measure pad to pad spacing)	0.18mm Good		Gerber\PCB_BLE_Speaker-F_Cu.gbr Gerber\PCB_BLE_Speaker-B_Cu.gbr	Danger: 0 Warning: 0 Good: 58
Plated through-hole to trace clearance (Measure clearance of plated through-holes to traces)	0.25mm Good		Gerber\PCB_BLE_Speaker-F_Cu.gbr	Danger: 0 Warning: 0 Good: 8
Annular ring (Annular ring width of pads compared to holes)	0.11mm Warning		Gerber\PCB_BLE_Speaker-F_Cu.gbr Gerber\PCB_BLE_Speaker-B_Cu.gbr	Danger: 0 Warning: 87 Good: 9
tht to smd (Detect clearance of vias to pads)	0.52mm Danger		Gerber\PCB_BLE_Speaker-F_Cu.gbr Gerber\PCB_BLE_Speaker-B_Cu.gbr	Danger: 12 Warning: 8 Good: 34
Via to pad (Detect clearance of vias to pads)	No	No	No	Danger: 0 Warning: 0 Good: 0

PCB DFM>Soldermask layer analysis

Analyze project	Analysis results	PCB screenshot	Layer distribution	Statistics
Soldermask bridge (Detect distance between parallel soldermask opening edges)	0.2mm Good		Gerber\PCB_BLE_Speaker-F_Mask.gbr	Danger: 0 Warning: 0 Good: 50
Solder mask opening exposing trace (Detect clearance of solder mask openings to nearby traces)	0.17mm Good		Gerber\PCB_BLE_Speaker-F_Mask.gbr	Danger: 0 Warning: 0 Good: 34
Soldermask opening with multiple segments (Check if solder mask openings are constructed from multiple geometric shapes)	null Warning		Gerber\PCB_BLE_Speaker-F_Mask.gbr	Danger: 0 Warning: 5 Good: 0
Negative soldermask expansion (Detect solder mask openings smaller than their corresponding pads)	0.12mm Warning		Gerber\PCB_BLE_Speaker-F_Mask.gbr	Danger: 0 Warning: 1 Good: 0

PCB DFM>Silkscreen layer analysis

Analyze project	Analysis results	PCB screenshot	Layer distribution	Statistics
Silkscreen to pad (Detect clearance of silkscreen to pads)	0mm Danger		Gerber\PCB_BLE_Speaker-F_Silkscreen.gbr	Danger: 39 Warning: 0 Good: 11
Silkscreen to hole (Detect clearance of silkscreen to holes)	0mm Danger		Gerber\PCB_BLE_Speaker-F_Silkscreen.gbr	Danger: 50 Warning: 0 Good: 0
Silkscreen line width (Check silkscreen line width)	0.1mm Danger		Gerber\PCB_BLE_Speaker-F_Silkscreen.gbr	Danger: 11 Warning: 39 Good: 0

PCB DFM>Drill layer analysis

Analyze project	Analysis results	PCB screenshot	Layer distribution	Statistics
Unconnected via (Detect isolated unconnected vias)	No	No	No	Danger: 0 Warning: 0 Good: 0
Missing plated through-hole (Detect top and bottom pads at the same location without plated through-holes)	No	No	No	Danger: 0 Warning: 0 Good: 0
Unconnected via (Detect isolated unconnected vias)	No	No	No	Danger: 0 Warning: 0 Good: 0
Plated through-hole spacing (Measure spacing between plated through-holes)	No	No	No	Danger: 0 Warning: 0 Good: 0
Short slot detection (Detect slots shorter than twice their width)	No	No	No	Danger: 0 Warning: 0 Good: 0
Slot width check (Measure slot width)	0.6mm Danger		Gerber\PCB_BLE_Speaker-PTH.drl	Danger: 4 Warning: 2 Good: 0
Via to PTH spacing (Measure spacing of vias to plated through-holes)	No	No	No	Danger: 0 Warning: 0 Good: 0
Unconnected via (Detect isolated unconnected vias)	No	No	No	Danger: 0 Warning: 0 Good: 0