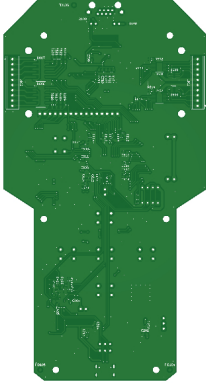
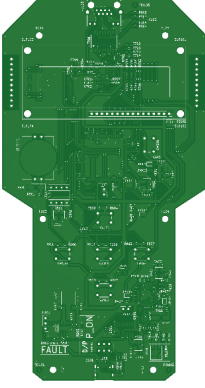
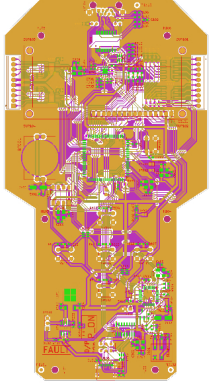


DFM analysis report_JLCDFM

File name :	Gerber.zip
Report generated at :	2025-10-28 10:29:01
PCB layers :	2
PCB size :	10x19cm
Analyze project :	<input checked="" type="checkbox"/> PCB DFM <input type="checkbox"/> 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)	No	No	No	Danger: 0 Warning: 0 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_Tarjeta_DevOps-F_Cu.gbr Gerber\PCB_Tarjeta_DevOps-B_Cu.gbr	Danger: 0 Warning: 0 Good: 100
Unconnected trace end (Free-standing trace ends not connected to pads)	No	No	No	Danger: 0 Warning: 0 Good: 0
Trace width (Trace width information)	0.2mm Good		Gerber\PCB_Tarjeta_DevOps-F_Cu.gbr Gerber\PCB_Tarjeta_DevOps-B_Cu.gbr	Danger: 0 Warning: 0 Good: 64
Fiducial (Detect fiducial marks on the board)	null Warning		Gerber\PCB_Tarjeta_DevOps-F_Cu.gbr Gerber\PCB_Tarjeta_DevOps-B_Cu.gbr	Danger: 0 Warning: 6 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_Tarjeta_DevOps-F_Cu.gbr Gerber\PCB_Tarjeta_DevOps-B_Cu.gbr	Danger: 0 Warning: 0 Good: 100
Plated through-hole to trace clearance (Measure clearance of plated through-holes to traces)	0.35mm Good		Gerber\PCB_Tarjeta_DevOps-F_Cu.gbr Gerber\PCB_Tarjeta_DevOps-B_Cu.gbr	Danger: 0 Warning: 0 Good: 36
Annular ring (Annular ring width of pads compared to holes)	0.15mm Warning		Gerber\PCB_Tarjeta_DevOps-F_Cu.gbr Gerber\PCB_Tarjeta_DevOps-B_Cu.gbr	Danger: 0 Warning: 100 Good: 0
tht to smd (Detect clearance of vias to pads)	No	No	No	Danger: 0 Warning: 0 Good: 0
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)	Unanalyzed	No	No	No
Solder mask opening exposing trace (Detect clearance of solder mask openings to nearby traces)	Unanalyzed	No	No	No
Soldermask opening with multiple segments (Check if solder mask openings are constructed from multiple geometric shapes.)	Unanalyzed	No	No	No
Negative soldermask expansion (Detect solder mask openings smaller than their corresponding pads)	Unanalyzed	No	No	No

PCB DFM>Silkscreen layer analysis

Analyze project	Analysis results	PCB screenshot	Layer distribution	Statistics
Silkscreen to pad (Detect clearance of silkscreen to pads)	No	No	No	Danger: 0 Warning: 0 Good: 0
Silkscreen to hole (Detect clearance of silkscreen to holes)	0mm Danger		Gerber\PCB_Tarjeta_DevOps-B_Silkscreen.gbr Gerber\PCB_Tarjeta_DevOps-F_Silkscreen.gbr	Danger: 90 Warning: 10 Good: 0
Silkscreen line width (Check silkscreen line width)	0.1mm Danger		Gerber\PCB_Tarjeta_DevOps-B_Silkscreen.gbr Gerber\PCB_Tarjeta_DevOps-F_Silkscreen.gbr	Danger: 18 Warning: 82 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)	null Warning		Gerber\PCB_Tarjeta_DevOps-F_Cu.gbr	Danger: 0 Warning: 6 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)	0mm Danger		Gerber\PCB_Tarjeta_DevOps-PTH.drl	Danger: 50 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_Tarjeta_DevOps-PTH.drl	Danger: 4 Warning: 0 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