

DFM analysis report_JLCDFM

File name: Gerber.zip Report generated at: 2025-10-28 10:29:01 PCB layers: PCB size: 10x19cm

✓ PCB DFM SMT DFM Analyze project:



(Detect clearance of vias to pads)

Analyze project Soldermask bridge

Silkscreen to pad

(Detect isolated unconnected

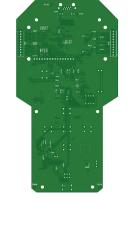
vias)

No

No

Analysis results





Good: 0 Danger: 0

Warning: 0

Good: 0

Statistics

Danger: 0

Analyze project		B DFM>Routing layer analy		Statistics
	Analysis results	PCB screensnot	Layer distribution	
Sharp trace corner				Danger: 0
(Check for sharp corners in	No	No	No	Warning: 0
traces)				Good: 0
				Danger: 0
BGA pad	No	No	No	Warning: 0
(Check BGA pads on the board)	140	No	140	Good: 0
				G00d · U
Via placed within a pad				Danger: 0
(Check if there is a via placed	No	No	No	Warning: 0
within a pad)				Good: 0
Trace to board edge				Danger: 0
=	NI-	N-	N-	_
(Detect traces too close to the	No	No	No	Warning: 0
board edge)				Good: 0
			Gerber\PCB_Tarjeta_DevOps-	
Trace spacing	0.2mm			Danger: 0
(Measure spacing between			F_Cu.gbr	Warning: 0
adjacent parallel traces)	Good		Gerber\PCB_Tarjeta_DevOps-	Good: 100
			B_Cu.gbr	
Unconnected trace end				Danger: 0
(Free-standing trace ends not	No	No	No	Warning: 0
connected to pads)	110			Good: 0
connected to pade/				G000+ 0
			Gerber\PCB_Tarjeta_DevOps-	
Trace width	0.2mm		F_Cu.gbr	Danger: 0
(Trace width information)	Good		Gerber\PCB_Tarjeta_DevOps-	Warning: 0
(Trade Width Information)	-		B_Cu.gbr	Good: 64
			5_0d.gs/	
			Gerber\PCB_Tarjeta_DevOps-	
Fiducial	null		F_Cu.gbr	Danger: 0
(Detect fiducial marks on the	Warning		Gerber\PCB_Tarjeta_DevOps-	Warning: 6
board)			B_Cu.gbr	Good: 0
Pad to board edge				Danger: 0
(Measure distance of pads from	No	No	No	Warning: 0
the board edge)				Good: 0
			Gerber\PCB_Tarjeta_DevOps-	
Pad spacing	0.18mm		F_Cu.gbr	Danger: 0
		_		Warning: 0
(Measure pad to pad spacing)	Good		Gerber\PCB_Tarjeta_DevOps-	Good: 100
			B_Cu.gbr	
Plated through-hole to trace			Gerber\PCB_Tarjeta_DevOps-	
clearance	0.35mm		F_Cu.gbr	Danger: 0
(Measure clearance of plated	Good		Gerber\PCB_Tarjeta_DevOps-	Warning: 0
through-holes to traces)			B_Cu.gbr	Good: 36
			5_54.95.	
Annular ring			Gerber\PCB_Tarjeta_DevOps-	Danger: 0
-	0.15mm		F_Cu.gbr	
(Annular ring width of pads	Warning		Gerber\PCB_Tarjeta_DevOps-	Warning: 100
compared to holes)			B_Cu.gbr	Good: 0
				Danger: 0
tht to smd	No	No	No	Warning: 0
Detect clearance of vias to pads)	140			Good: 0
				L=OOd: D

No

PCB DFM>Soldermask layer analysis

PCB screenshot

No

Layer distribution

Analyze project	Analysis results	PCB screenshot	Layer distribution	Statistics	
PCB DFM>Silkscreen layer analysis					
Negative soldermask expansion (Detect solder mask openings smaller than their corresponding pads)	Unanalyzed	No	No	No	
Soldermask opening with multiple segments (Check if solder mask openings are constructed from multiple geometric shapes)	Unanalyzed	No	No	No	
Solder mask opening exposing trace (Detect clearance of solder mask openings to nearby traces)	Unanalyzed	No	No	No	
(Detect distance between parallel soldermask opening edges)	Unanalyzed	No	No	No	

(Detect clearance of silkscreen to	No	No	No	Warning: 0		
pads)				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 <mark>Danger</mark>		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				Danger: 0		

Analyze project	Analysis results	PCB screensnot	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				Danger: 0

No

No

Warning: 0 Good: 0