

DFM analysis report\_JLCDFM

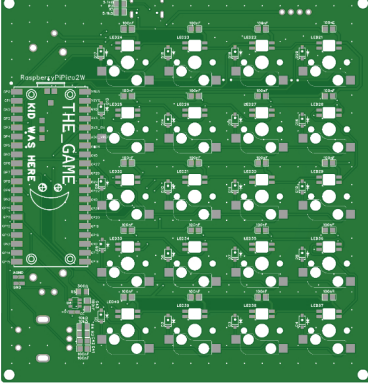
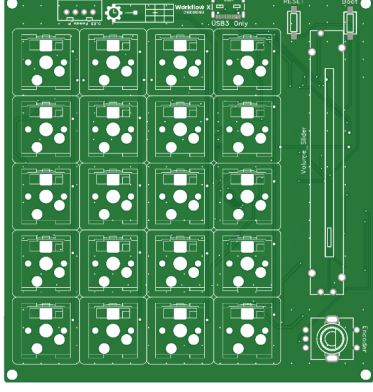
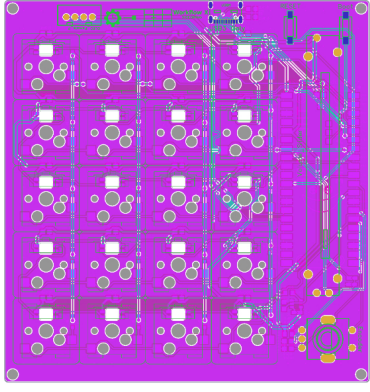
File name: WorkflowX\_Final.zip

Report generated at: 2025-10-21 09:58:35

PCB layers: 2

PCB size: 10.5x11cm

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)	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)	0.15mm Warning		Gerber_BottomLayer.GBL Gerber_TopLayer.GTL	Danger: 0 Warning: 7 Good: 2
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)	No	No	No	Danger: 0 Warning: 0 Good: 0
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)	0.23mm Good		Gerber_BottomLayer.GBL Gerber_TopLayer.GTL	Danger: 0 Warning: 0 Good: 52
Pad spacing (Measure pad to pad spacing)	0.14mm Warning		Gerber_BottomLayer.GBL Gerber_TopLayer.GTL	Danger: 0 Warning: 1 Good: 18
Plated through-hole to trace clearance (Measure clearance of plated through-holes to traces)	0.26mm Good		Gerber_BottomLayer.GBL Gerber_TopLayer.GTL	Danger: 0 Warning: 0 Good: 100
Annular ring (Annular ring width of pads compared to holes)	0.15mm Warning		Gerber_BottomLayer.GBL Gerber_TopLayer.GTL	Danger: 0 Warning: 100 Good: 0
tht to smd (Detect clearance of vias to pads)	1.39mm Danger		Gerber_BottomLayer.GBL Gerber_TopLayer.GTL	Danger: 9 Warning: 4 Good: 45
Via to pad (Detect clearance of vias to pads)	0.09mm Danger		Gerber_BottomLayer.GBL	Danger: 2 Warning: 2 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)	0.1mm Danger		Gerber_TopSolderMaskLayer.GTS	Danger: 11 Warning: 0 Good: 0
Solder mask opening exposing trace (Detect clearance of solder mask openings to nearby traces)	0.09mm Warning		Gerber_BottomSolderMaskLayer.GBS Gerber_TopSolderMaskLayer.GTS	Danger: 0 Warning: 56 Good: 0
Soldermask opening with multiple segments (Check if solder mask openings are constructed from multiple geometric shapes)	null Warning		Gerber_BottomSolderMaskLayer.GBS	Danger: 0 Warning: 6 Good: 0
Negative soldermask expansion (Detect solder mask openings smaller than their corresponding pads)	No	No	No	Danger: 0 Warning: 0 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_BottomSilkscreenLayer.GBO Gerber_TopSilkscreenLayer.GTO	Danger: 100 Warning: 0 Good: 0
Silkscreen to hole (Detect clearance of silkscreen to holes)	0mm Danger		Gerber_BottomSilkscreenLayer.GBO Gerber_TopSilkscreenLayer.GTO	Danger: 100 Warning: 0 Good: 0
Silkscreen line width (Check silkscreen line width)	0.06mm Danger		Gerber_BottomSilkscreenLayer.GBO Gerber_TopSilkscreenLayer.GTO	Danger: 50 Warning: 50 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		Drill_PTH_Through.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