*SubsystemDesign — PCB Editor

File Edit View Place Route Inspect Tools Preferences Help

Track: use netclass width | Via: use netclass sizes | F.Cu (PgUp) | 0.2000 mm (7.87 mils) | Zoom 1.50

Design Rules Checker

Refill all zones before performing DRC
Test for parity between PCB and schematic
Report all errors for each track

Violations (0) | Unconnected Items (0) | Schematic Parity (not run) | Ignored Tests (5)

Show: All | Errors 0 | Warnings 0 | Exclusions | Save...

Delete Marker | Delete All Markers | Run DRC | Close

Objects Nets

Cu
Cu
Adhesive
Adhesive
Paste
Paste
Silkscreen
Silkscreen
Mask
Mask
User.Drawings
User.Comments
User.Eco1
User.Eco2
Edge.Cuts
Margin
Courtyard
Courtyard
Fab
Fab
User.1
User.2

Layer Display Options

Presets (Ctrl+Tab): ---

Viewports (Shift+Tab): ---

Selection Filter

All items | Locked items
Footprints | Text
Tracks | Vias
Pads | Graphics
Zones | Rule Areas
Dimensions | Other items

Pads 136 | Vias 0 | Track Segments 87 | Nets 73 | Unrouted 0

File 'C:\Users\16238\OneDrive\Desktop\SubsystemDesign\SubsystemDesign.kicad_pc... | Z 1.44 | X 26.8000 Y 58.4000 | dx 26.8000 dy 58.4000 dist 64.2557 | grid 0.2000 | mm | Select item(s)