



General Notes

1. Fabricate per IPC-610-A Class 2, ANSI/IPC-A-600, IPC-QE-605, IPC-4101, IPC-4552 and IPC-SM-840 Specifications. Use of EU ROHS-Compliant materials required. EU REACH compliance preferred but not required.
2. Material: Base material laminated epoxy glass FR-4, nominal core thickness 0.062" / 1.54mm, color natural, nominal 1oz copper weight. Thickness 0.07" / 1.62mm max after plating and finishing. ROHS compliant per IPC-4101 slash sheets #26 or #83 or #98 with minimum Tg 130 Degree C or Higher, Td 300 Degree C or Higher & Flame Rated UL 94V-0.
3. Soldermask both sides of board over bare copper with material per ANSI/IPC-SM-840. Color shall be blue.
4. Apply Electroless nickel Immersion gold plating (ENIG) to exposed copper on both sides.
5. Bottom side silkscreen available to apply markings such as manufacturer, UL marking, date-code, etc. Avoid modifying top side silkscreen with manufacturer information, UL marking, datecode, etc.
6. Apply silkscreen to top & bottom side of board using non-conductive white epoxy ink.
7. See fabrication files with same title or drawing number and revision as this drawing.
8. Dimensional tolerances are: 0.XX" = +/- 0.01", 0.XXX = +/- 0.005, angles = +/- 1 degrees.
9. Outline defined in separate gerber file with ".GM1" file extension.
10. See separate drill file with ".DRL" file extension for hole locations. Selected hole locations in this drawing are for reference only.
11. Hole locations specified in the separate drill file take precedence over this drawing and artwork. Contact purchaser / engineer to resolve dimension conflicts between drawing and artwork, if any.
12. Hole & slot sizes are specified as final dimensions after plating and finishing unless otherwise specified.
13. Vendor to plate holes and exposed pads.
14. After assembly, trim solder fillets and component leads to 0.06" (Max) beyond board surface on bottom side.
15. Design geometry minimum feature sizes & design rules:
 - Trace Width: 6mils
 - Hole-Hole: 25mils
 - Silkscreen Line Width: 6mils
 - Minimum Hole Size (Plated): 10mils
 - Board Edge-Copper: 15mils
 - Trace-Trace / Trace-Pad: 6mils
 - Pad-Pad: 6mils
 - Silkscreen-Bare Copper: 6mils
 - Maximum Hole Size (Unplated): 250mils
16. Panelization to be done by the fab house. Mouse bites and routing is acceptable. V-score is not acceptable.

Stackup:

F.Mask - Top Mask - 1mil thick max
F.Cu - Top Layer - 1oz Copper

FR-4 Dielectric 62mils

B.Cu - Bottom Layer - 1oz Copper
B.Mask - Bottom Mask - 1mil thick max

Design by Lambda Wolf Labs.



Drawing Number:
LWL-003-E10

Lambda Wolf Labs (@LorneChrones)

Sheet:
File: FWA_2022_Compass_Badge_LWL-003-E10.kicad_pcb

Title: FWA 2022 Compass Badge

Size: A4 Date: 2022-02-12

KiCad E.D.A. kicad (5.1.2)-2

Rev: B

Id: 1/1