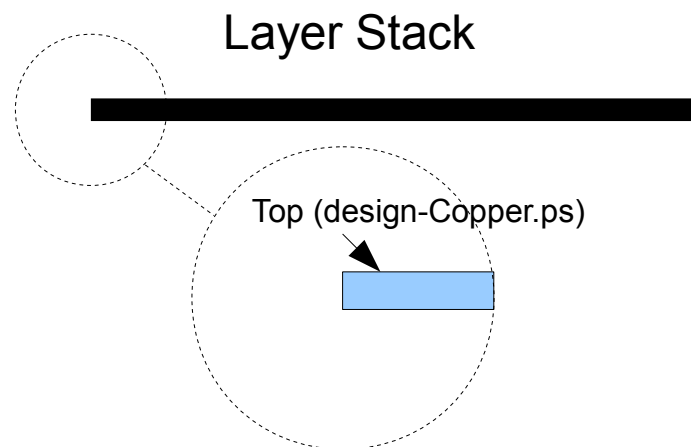


- T1 0.032" 0.81mm (14 holes)
- T2 0.040" 1.02mm (3 holes) (with 3 oblongs)
- T3 0.050" 1.27mm (2 holes)
- T4 0.120" 3.05mm (2 holes)



| SWIFT CONSTRUCTION COMPANY | | | |
|----------------------------|-----------------------|--------|---------------|
| PRJ | Aircraft Wireless | | |
| TITLE1 | PCB,AIRCRAFT WIRELESS | | |
| TITLE2 | Master Drawing | | |
| DWG# | 20000002 | REV 0 | CAT MA |
| DATE | 2013-01-02 | DRN BJ | PAGE 01 OF 02 |

FABRICATION NOTES

1. General Characteristics

Size: 3.30" x 2.00"

Layers: 1

Holes:

- T1 0.032" 0.81mm (14 holes)
- T2 0.040" 1.02mm (3 holes) (with 3 oblongs)
- T3 0.050" 1.27mm (2 holes)
- T4 0.120" 3.05mm (2 holes)

2. Board shall be fabricated to meet:

- IPC-6012, Class 2 "Qualification and Performance Specifications for Rigid PCBs"
- IPC-A-600, Class 2 "Acceptability of Printed Boards"

3. Electrical testing is not required.

4. Material:

- FR4 per IPC 4101/21.
- Copper cladding for external layers to be 2 ounce copper before plating.
- Overall board thickness to be 0.062" +/- 0.003" and is inclusive of all plating and solder mask coatings.

5. Plating:

- All plated through holes shall be copped plated to a nominal thickness of 0.0015" and have a minimum thickness of 0.0012".
- All exposed copper shall be tin-lead plated and fused or solder coated to a minimum thickness of 0.0003" per IAW QQ-S-571.

6. Solder mask:

- Liquid photo-imageable green solder mask shall be applied over bare copper on both sides of the board per IPC-SM-840, Class H.
- Solder mask registration shall be within 0.005" and have a minimum thickness of 0.001".

7. Legend:

- White non-conductive epoxy ink conforming to A-A-56032 specifications shall be applied to the component side of the board.
- Legend ink shall not be permitted on exposed pads or in holes.

9. Process Allowances:

- Conductor width shall not vary by more than +/-0.002".
- Conductor spacing shall not be less than 0.010".
- Plated through holes shall have a minimum annular ring of 0.005".
- Dimensional tolerance for all holes is +/-0.005".
- Positional tolerance for all holes is +/-0.005".
- Front to back registration shall not vary by more than +/- 0.005".
- Maximum warp and twist across diagonal corners of the board shall not exceed 0.0075" per inch.

10. All layers including the drill and trim views are shown as viewed through the board from the component side.

11. Panelisation. This drawing defines the pre-panelised board. Where panelisation are required, a separate drawing will define panelisation.

12. Vendor marking. Vendor is to mark boards with Vendor ID and Date Code in convenient area.

13. This drawing is to be used in conjunction with PCB artwork file: 20000002_CAM.ZIP (Revision 1). Gerber file format is RS-274-X, Format 3.4, Imperial, Leading Zero Omitted, Absolute. Drill file format is EXCELLON.

SWIFT CONSTRUCTION COMPANY

PRJ Aircraft Wireless

TITLE1 PCB,AIRCRAFT WIRELESS

TITLE2 Master Drawing

| | | | | | |
|------|------------|-----|----|------|----------|
| DWG# | 20000002 | REV | 0 | CAT | MA |
| DATE | 2013-01-02 | DRN | BJ | PAGE | 02 OF 02 |