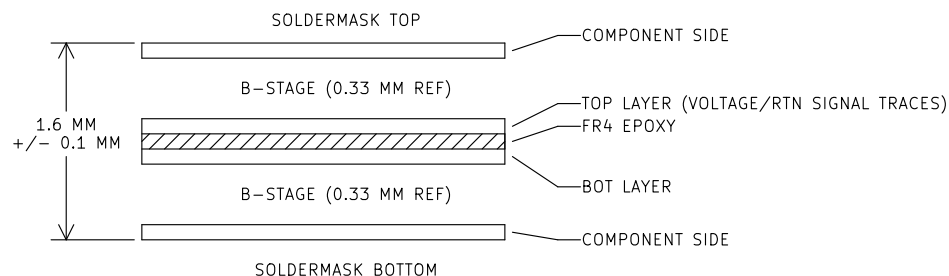


Drill Map:

- 0.50mm / 0.020" (8 holes)
- 0.60mm / 0.024" (6 holes)
- 0.80mm / 0.031" (80 holes)
- 0.90mm / 0.035" (6 holes)
- 1.00mm / 0.039" (4 holes + 3 slots)
- 1.10mm / 0.043" (13 holes)
- * 3.50mm / 0.138" (4 holes) (not plated)

LAYER DETAIL



GENERAL NOTES

1. FABRICATE PER ANSI/IPC-1-600, IPC-QE-605, IPC-4101, IPC-4552 AND IPC-SM-840 SPECIFICATIONS. MUST USE ROHS-COMPLIANT MATERIALS.
2. MATERIAL: BASE MATERIAL LAMINATED EPOXY GLASS FR-4, NOM .062", COLOUR NATURAL. NOM 1 OZ COPER WEIGHT. THICKNESS .070" MAX AFTER PLATING AND FINISHING. PREFER ROHS COMPLIANT PER IPC-4101 SLASH SHEETS #26 OR #83 OR #98 WITH MINIMUM TG 135 DEGREE C OR HIGHER, TD 300 DEGREE C OR HIGHER AND FLAME RATED UL 94V-0.
3. SOLDERMASK BOTH SIDES OF BOARD OVER BARE COPPER WITH MATERIAL PER ANSI/IPC-SM-840, COLOR SHALL BE GREEN.
4. APPLY SOLDER PLATING WITH HOT-AIR LEVELLING TO EXPOSE COPPER BOTH SIDES. LEAD-FREE SOLDER REQUIRED.
5. INDICATED AREA ON BOTTOM SIDE AVAILABLE FOR APPLICATION OF PCB MANUFACTURER LOGO, UL MARKING, AND DATE-CODE. USE OF OTHER LOCATION TO BE APPROVED BY PURCHASER.
6. APPLY SILKSCREEN TO TOP 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. DIMNESONAL TOLERANCES ARE: .XX=+/- .25 MM; .XXX=+/- .127 MM.
9. OUTLINE DEFINED IN SEPERATE GERBER FILE. DIMENSIONS OF CIRCUMSCRIBED RECTANGLE SHOWN ON THIS DWG FOR REF ONLY.
10. SEE SEPERATE DRILL FILE FOR HOLE LOCATIONS. SELECTED HOLE LOCATIONS SHOWN ON THIS DWG FOR REF ONLY.
11. HOLE LOCATIONS SPECIFIED IN SEPERATE DRILL FILE TAKE PRECEDENCE OVER THIS DWG AND ARTWORK. CONTACT PURCHASER TO RESOLVE DIMENSION CONFLICT BETWEEN DWG AND ARTWORK.
12. HOLE SIZES ARE SPECIFIED AS FINAL DIMENSIONS AFTER PLATING AND FINISHING. UNLESS OTHERWISE SPECIFIED ALL HOLES TO BE PLATED.
13. VENDOR TO PLATE HOLES AND EXPOSED PADS. VENDOR TO SPECIFY TYPE OF PLATING MATERIAL TO BE USED WHEN ACCEPTING ORDER.
14. AFTER ASSEMBLY TRIM SOLDER FILLETS AND COMPONENT LEADS TO 1.5 MM (MAX) BEYOND BOARD SURFACE ON BOTTOM SIDE.
15. DESIGN GEOMETRY MINIMUM FEATURE SIZES:

TRACE WIDTH; ANULUS	0.50 MM	TRACE-TO-TRACE; TRACE-TO-PAD	0.20 MM
HOLE-TO-HOLE	1.50 MM	PAD-TO-PAD	0.20 MM
SILKSCREEN LINE	0.20 MM	SILKSCREEN-TO-BARE COPPER	0.20 MM
MINIMUM HOLE SIZE (PLATED)	0.60 MM	MAXIMUM HOLE SIZE (UNPLATED)	3.50 MM
BOARD EDGE-TO-COPPER	0.50 MM		

Drawn by: Andrew Li

Sheet:

File: audio-amplifier.kicad_pcb

Title: Audio Amplifier

Size: A4 Date: 2020-05-19

KiCad E.D.A. kicad (5.1.5-0-10_14)

Rev: B

Id: 1/1