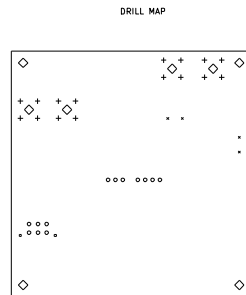
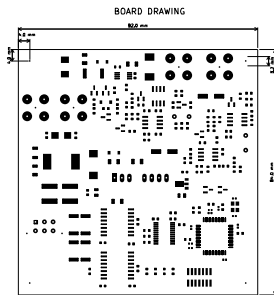


REVISION HISTORY		
REV	AUTHOR	NOTES
A	J. McNeilly	Initial release.



Drill Map:

* 0.800mm / 0.0315"	(4 holes)
o 1.000mm / 0.0394"	(13 holes)
+ 1.850mm / 0.0728"	(16 holes)
e 1.000mm / 0.0394"	(2 holes) (not plated)
o 3.200mm / 0.1260"	(8 holes) (not plated)

NOTES: UNLESS OTHERWISE SPECIFIED.

1. MATERIAL: FRA T74 150 C OR EQUIVALENT.
2. EQUIVALENT MATERIAL SHALL BE RHPs COMPLIANT, HALOGEN FREE AND APPROVED BY PARTS FOR BIRDS.
3. C. THICKNESS OF INDIVIDUAL COPPER CLAD SHEETS SHALL BE AS IN DEFINED IN STACK-UP.
4. ETC.
5. GEOMETRY:
 - A. MEASURE WIDTH FROM THE BASE OF THE METALLIZATION.
 - B. MINIMUM LINE WIDTH: 0.15 MM OUTER, 0.15 MM INNER LAYERS.
 - C. MINIMUM LINE WIDTH AND TERMINAL AREA: 0.15 MM. DEVIATE FROM THE 1-TO-1 MASTER PATTERN IMAGE BY MORE THAN ± 0.025 MM OR 0.2X, WHICHEVER IS LESS.
6. SURFACE FINISH:
 - A. ENIG PLATING PER CURRENT REVISION OF IPC-4552. EXPOSED METAL SHALL HAVE 118-236 MICRON INCHES THICKNESS AND 118-236 MICRON INCHES DOWNS.
 - B. IMPEDANCE (ALL TOLERANCES \pm 10%).
 - C. ALL 0.2 MM WIDE/0.15 MM SPACE PARTS ON OUTER LAYERS SHALL BE 90 OHMS.
 - D. IMPEDANCE OF 0.2MM/0.15 MM SPACE PAIRS ON INNER LAYERS SHALL BE 100 OHMS.
 - E. VENDOR MAY ADJUST DESIGN GEOMETRIES UP TO ± 0.2 MM TO ACHIEVE TARGET IMPEDANCE. ADJUSTMENTS SHALL BE IN WIDTH, SPACING OR DIELECTRIC THICKNESS SHALL REQUIRE APPROVAL FROM PARTS FOR BIRDS ENGINEERING.
7. HOLES:
 - A. PLATING IN HOLES SHALL BE CONTINUOUS ELECTROLYTIC COATING WITH 0.25 MM MINIMUM BARREL THICKNESS.
 - B. MINIMUM FINISHED HOLE SIZE: 0.20 MM
 - C. HOLE SIZE MEASURED AFTER PLATING.
 - D. SEE DRILL CHART FOR FINISHED HOLE SIZE AND TOLERANCE.
 - E. ALL HOLES SHALL BE LOCATED WITHIN 0.08 MM OF TRUE POSITION AS SHOWN IN CAD DATA.
8. SOLDERMASKS:
 - A. SOLDERMASK OVER BAR COATING (SMOBC) ON PRIMARY AND SECONDARY SIDES USING SUPPLIED ARTWORK IN CONFORMANCE WITH CURRENT REVISION OF IPC-4552, CLASS 2.
 - B. COLOR: MATT GREEN
 - C. CURRENTLY UNIMAGEABLE (LPI) 0.001 MM TO 0.002 MM THICKNESS, HALOGEN FREE
 - D. NO BLEED-OUT ALLOWED OVER EXPOSED SMD PADS.
 - E. NO EXPOSED STROES.
9. SILKSCREENING:
 - A. SILKSCREEN PRIMARY AND SECONDARY SIDE WITH WHITE EPOXY, NON-CONDUCTIVE, NON-NUTRIENT INK.
 - B. UNINSPECTED STROKE WIDTH SHALL BE 0.13 MM
 - C. CLIP SILKSCREEN AWAY FROM ANY EXPOSED METAL.
 - D. ADDITIONAL COPY ADDITIONAL MARKING TO BE LOCATED ON THE SECONDARY SIDE.
 - E. BAG AND TAG ACCEPTABLE FOR PCBs THAT ARE TOO SMALL FOR MARKING.
10. STANDARDS:
 - A. FABRICATE PCB IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-6012, CLASS 2.
 - B. INTERPRET DIMENSIONS AND TOLERANCES IN ACCORDANCE WITH THE CURRENT REVISION OF ASME Y14.5M.
 - C. DO NOT EXCEED TOLERANCES.
11. FINISHES:
 - A. BOTH END TWIST OF ASSEMBLY SUB-PANEL OR SINGULATED PCB SHALL NOT EXCEED .025 MM PER MM.
 - B. TEST IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-TH-650.2.4.2.2
12. DESTRUCTIVE TESTING:
 - A. SOLDER SAMPLE PROCESSED THROUGH SOLDERING SHALL BE INCLUDED WITH EACH SHIPMENT.
 - B. X-OUT PANELS MAY BE USED FOR SOLDER SAMPLE.
 - C. REMOVE ALL BURRS AND BREAK SHARP EDGES 80,003 MM
13. NON-DESTRUCTIVE EVALUATION:
 - A. ALL PCBs SHALL PASS 100% ELECTRICAL TEST IN ACCORDANCE WITH IPC-356 NETLIST IN ACCORDANCE WITH CURRENT REVISION OF IPC-9252, CLASS 2.
 - B. CERTIFICATE OF CONFORMANCE SHALL BE SUPPLIED WITH EACH SHIPMENT.
14. X-OUTS:
 - A. X-OUT BOARDS THAT DO NOT MEET ALL SPECIFICATIONS USING PERMANENT MARKING ON BOTH SIDES OF THE EFFECTIVE PCB.
 - B. PANELS THAT DO NOT HAVE ANY X-OUTS SHALL BE PACKAGED TOGETHER.
 - C. PANELS THAT HAVE 0 OR FEWER X-OUTS SHALL BE PACKAGED SEPARATE FROM NON-X-OUT PANELS.
 - D. PANELS WITH MORE THAN 1 X-OUT SHALL BE PACKAGED SEPARATE FROM NON-X-OUT PANELS.
15. PACKAGING REQUIREMENTS:
 - A. PCBs SHALL BE PACKAGED IN VACUUM SEALED INVENT CONTAINERS.
 - B. OUTER CONTAINERS SHALL BE SUFFICIENT TO PREVENT DAMAGE DURING SHIPPING AND HANDLING.

STACKUP			
Layer Name	Material	Thickness	Dielectric Cons.
F.Cu	Copper	35um	
	FR4	0.11mm	4.29
In1.Cu	Copper	35um	
	FR4	1.13mm	3.96
In2.Cu	Copper	35um	
	FR4	0.11mm	4.29
B.Cu	Copper	35um	

John McNetty - jhallmcnetty@gmail.com
Pants for Birds
 Sheet:
 Files: high_side_switch_kicad_pcb
Title: High Side Switch

Size: C	Date:	Rev: A
Kicad 1.8.0, kicad (6.0.9)		ide 1/1