

Circuit design principles for Scout electronics projects for JOTA/JOTI

Aim

- To make simple circuits from discrete components and single function integrated circuits, so that Scouts can learn that the circuits are just composed of simple building blocks that can be easily understood.
- Design circuits that are both useful and appealing to the youth members
- Circuits must be able to withstand the assembly, and subsequent handling. (it must be tough)

Issues

- Soldering and assembly Skills of youth members learning to solder can be rather “harsh” on PCB's. 10-20% failure is not uncommon.
- Some components can be fragile. (physical weakness and static issues)
- Commercially available kits tend to have the battery, Speaker, switches and other components separate from the PCB - the end result is that the leads tend to be broken off speakers and batteries easily.

PCB Design principles

1. All components must be mounted on 1 board - e.g speaker, battery etc.
2. Use only most commonly available components
3. Special components that have limited availability, must have different options available, and the board must be designed to accept a number of alternatives if these are not available.
4. Circuits should use as little power as possible to extend battery life.

PCB Design

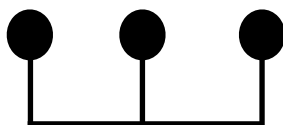
1. IC's - all mounted on sockets - IC's only inserted after soldering is checked, also ensures that IC's are not damaged though too much heat or static.
2. Battery holders - PCB mounted, but also attached by double sided tape to take the strain off the pins. Must be able to take different pin spacing from different manufacturers. Also be able to take a battery holder with wires (separate holes for this if necessary)
3. Speakers - PCB mount preferable - encased with pins, or bare speaker with leads. provision must be made to take different size speakers with different mountings. Attachment, solder in, or stick on (small mylar speakers tend to have a protruding ring that makes it easy to stick them on), also provision for screw holes, or even cable tie attachment.
4. On/Off Switches - preferable to Scouts just removing the battery. Switches must be anchored to the board by their casing as well as their leads if possible. Cases must be 1 piece, not spot welded together, so they do not break apart. Switch toggle must not extend past the board boundary, so there is less risk of it breaking off.

Momentary switches are preferable, so that circuits do not accidentally get left on and drain the battery. If Momentary switches are not suitable, then a power on led should be used as indication that the power has not been turned off.

5. Silk Screen - Front of PCB - component labelling, orientation “+ and -“, component outline, all special solder pads labelled - eg battery pcb mount, battery wire mount.
6. Silk Screen - Back of PCB - Large white space for Scouts name and group name. Revision number, circuit name
7. Lanyard attachment hole (5mm) - at the top of the board
8. Larger solder pads (2.5 to 3mm) - to better handle heat dissipation from excessive solder and extended heating times, and make it easier to solder.
9. Wider tracks (min 0.75mm) - in case pads are delaminated (common), so that component leads can be bent over solder mask scraped away and lead soldered to the remaining track - saves replacing the whole board and starting again.
10. Track Layout - last section of track to plated through holes is always on the solder side so that if a component completely covers the holes (eg pcb mount speaker or battery holder), and the pad is delaminated, the pin can still be connected to the track (see step 9).
11. Pin spacing - leave enough space between component ends and through holes so that the leads are not bent close to the component, this also allows for different value components - eg $\frac{1}{2}w$ if $\frac{1}{4} w$ resistors are not available
12. Room for components - ensure that all components have enough room around them to make assembly easy.
13. Axial leads components where possible as there is more spacing between pins on the board - so less likelihood of bridging holes.
14. Switches - connect all pins if they have connection to other pins, so if a track is delaminated then the switch will still work.
15. Only one track to pad connection, so that if a pad is delaminated then only one join has to be made



If the middle pad is delaminated then you need to make 2 connections - not preferable



If the middle pad is delaminated then you need to make only 1 connection - much easier to fix.

Layout summary

Pad sizes

1. Oversize pads - set to min 3 mm round on axial components
2. if pads close together then 2.5mm* 3 mm oval or round shape
3. Ic pads 2.5mm x 1.5mm oval

Track sizes and layout

1. 0.75mm tracks - to handle heat dissipation better, and allow for easy lead connection if pad is delaminated
2. Move tracks as far away from other holes as possible
3. Last section of track to through hole to be on component side.
4. 1 track to 1 pad, T join tracks instead, so that if a pad is delaminated there is only 1 connection to make.

Customisable Screen print suggestions

1. Scouts State Logo
2. Region Logo - if available)
3. Scout Group - name
4. Group website URL
5. Event details - JOTA/Jamborette and year if applicable
6. On Back - large white space for Name and also for Group/section

Contact - Venturers@cobargoscouts.org.au