

## DIY PCB's For Hammies Projects

During the lockdown, to keep myself busy and to keep out of the XYL's hair, I decided to look at a few simple electronic projects for the Hammies to build. One of the projects the XYL came up with was a 20 second timer. This is just a fun project for the kids to time the washing of hands. This could also be adapted to use as an egg timer or for board games.

Another project I did was a day/night sensor which the kids had lots of fun building. These kits were also used at the SARL stand at Hobby-X in March. In the past I always used Vero board for the projects. However, this was not always easy for the kids to follow and was often a frustrating process. Generally, in assembling the project, instructions caused confusion and many errors were made. Often, a substantial amount of time was spent fault finding and correcting the assembly. Therefore, I decided to take this opportunity, while at home under the lockdown, to start manufacturing my own PCB's.

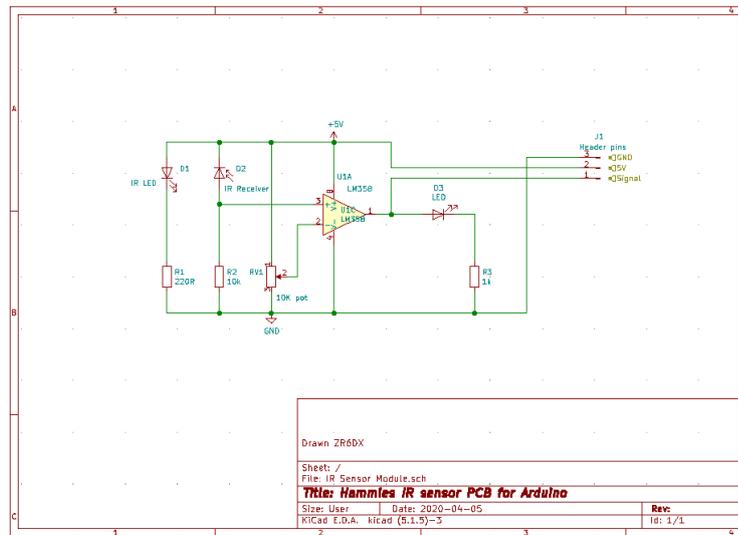
To manufacture your own boards, you would first need to layout the circuit in one of the many freely available CAD programs. Examples of suitable CAD programs are KiCad, Fritzing, FreeCad, ExpressPCB and even an online programme called EasyEDA. There are many other programs available if you Google "PCB Cad software" and make a selection.

Over a year ago I started to teach myself KiCad, which is an extremely powerful CAD program for electronic circuit design, and conversion to PCB layout. KiCad pronounced, "Key-CAD", is an open source program, and there are no restrictions as to board sizes or the applicable number of layers.

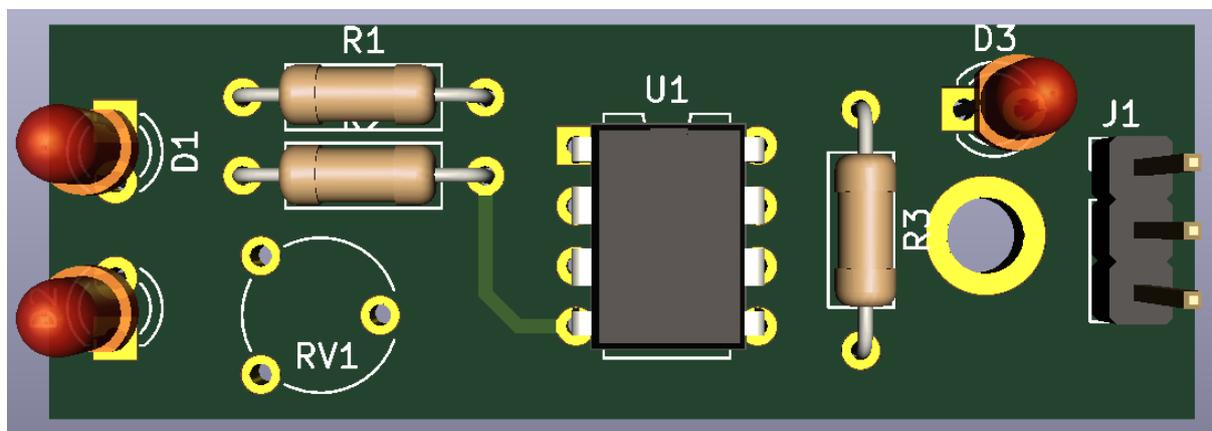
There are many methods used in home brewing PCB's, but I don't like fiddling around and wanted a method that was reliable that would produce consistent results. So, here I reveal my tried and tested method. The tools and material I use are as follows:

- Software KiCad
- B&W **laser** printer.
- Single sided or double-sided copper board depending on the project.
- Ferric Chloride for etching the copper.
- Press-n- Peel PCB transfer film. (sold by Communica)
- Steel wool for cleaning the copper board.
- "Brillo" pad, "Scotch Brite" scourer or similar used with soap to clean the copper board.
- XYL's iron and ironing board. Ask for instructions on how to use this household item. 😊
- 0.8mm to 1.2mm drills and a 3mm drill used for mounting holes.
- Dremel drill and drill press or similar.
- Dremel cutting disks (used to cut the PCB to the correct size before etching, less copper to etch.

First, you need to draw your circuit. The circuit diagram below, is an IR sensor board using a LM358 op-amp. This board is going to be used as the sensor for the Hammies' line follower car, which uses an Arduino Microprocessor. This future project should prove to be an exciting challenge for the youngsters, as they learn about electronics and Arduino programming. The board layout has been designed for THT components, as this makes it easier for the kids to build, and no special soldering equipment is required. Please feel free to use this circuit, if you wish. There are many similar designs on the Internet.

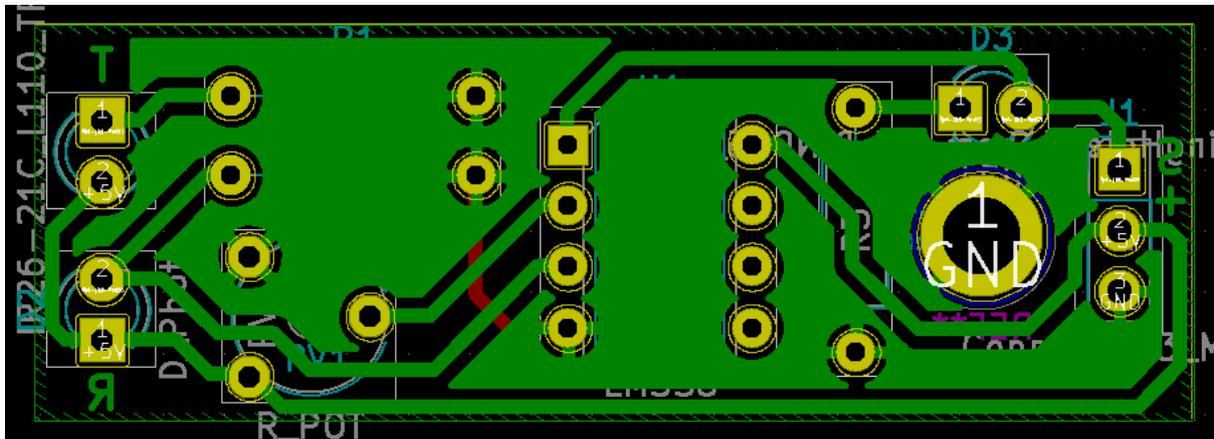


The layout of the PCB is critical as in most cases, it is much easier to homebrew a single-sided PCB than a double-sided PCB. It is preferable to get all the tracks on one side without having to include any jumpers or bridges on the top side of the board. We also want to make the board as neat and as small as possible. This is to ensure that we do not waste valuable etching products and PCB material. Below is the layout in 3D of the IR sensor PCB.

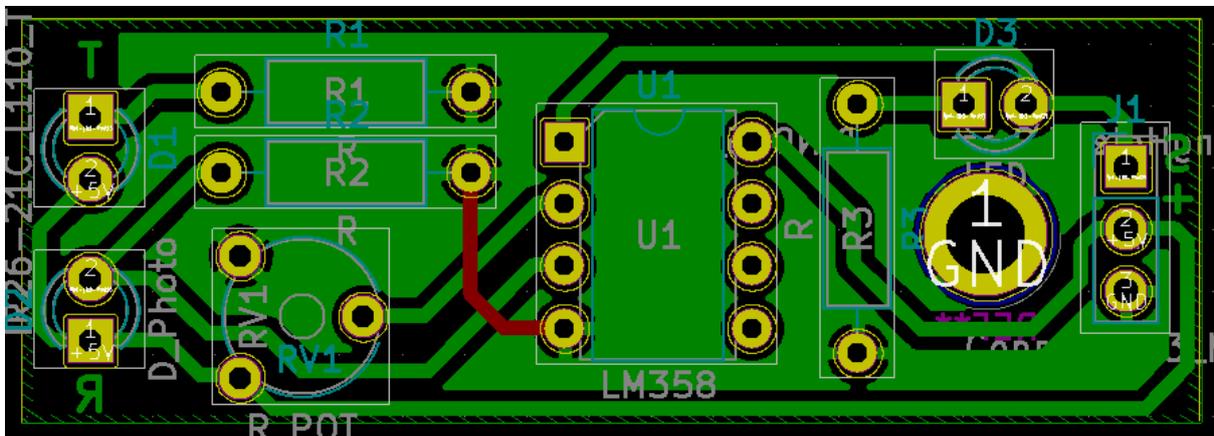


The actual PCB size is 16mm x 48mm, making it a small PCB. Before we look at the actual layout, we need to understand what we want to achieve. i.e. we want all the components on the top and all the copper tracks on the reverse side of the board. This is a simple process when using CAD software as you are able to select the layer you are working on. When working on a single layer board, I always layout the tracks on the back of the board. The reason for this is that you can just print the layout directly on top of the transfer film, and then place this on the copper layer. There is no need to mirror/reverse the image before printing. Your components will then fit to the PCB as you have designed. In Kicad the top layer colour is red, and the bottom layer colour is green. For KiCad use the “V” key to switch between layers when laying out the tracks. The “V” in KiCad actual means “via”.

Below is the image of the back of the board. Note the one red line. This is a jumper that I needed to add to get all the ground planes to join. As you will see from the image this is the bottom view of the layout.



The image below is the layout from the top of the board, note that the red track at the top of the board. From the image you can also see the component mask giving the annotation of each component.

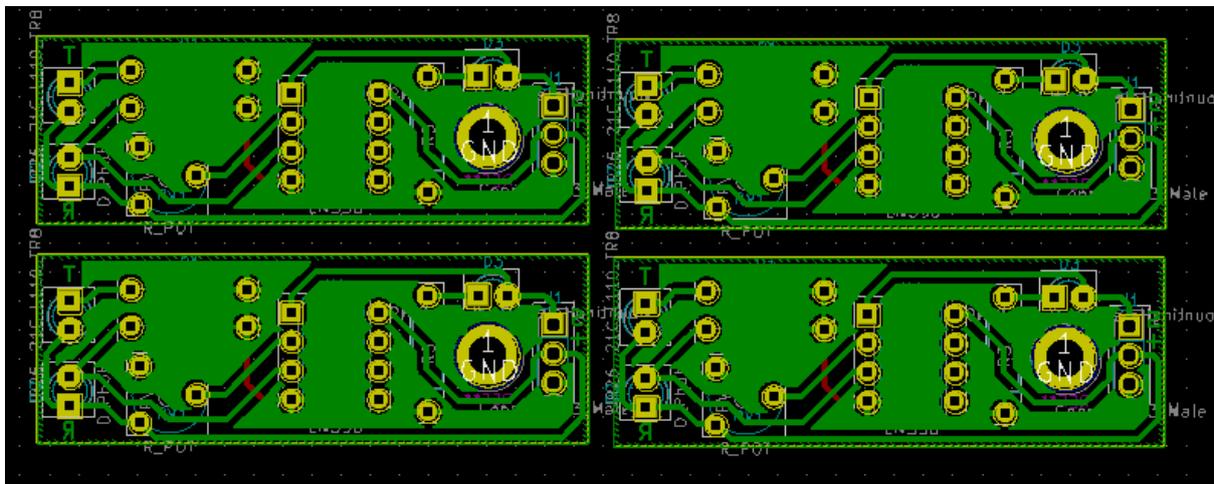


Kindly note that before printing the layout, make sure you have selected the bottom of the PCB. You also need to de-selected all the other technical layers such as the masks and silk layers. All writing needs to be mirrored. See the mirrored letters on the above PCB. I am sure you have noticed that I have included a ground plane. There are two reason for this: - the first one is that I don't have to place the ground tracks to and from each component; and the second reason is that when etching the board there is not a lot of copper to etch. This also makes the Ferric Chloride last a lot longer.

When laying out tracks, it must be noted that our tracks that supply current should be wide enough to handle the current. It is futile making a track 0.3mm wide, when that track needs to carry a current of a few amps, therefore, care must be taken when planning the layout. In most of the CAD

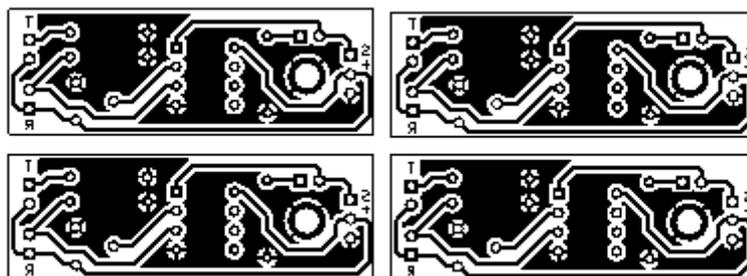
programs you can stipulate the Net-Class i.e. the tracks widths beforehand. So, when selecting a track that supplies current it will automatically select the track width that you predefined. In the above image it can be seen that on the board all the tracks are 0.8mm wide and the spacing is 0.2mm wide. The 0.8mm tracks widths are more than sufficient to handle the current. You must also remember that the narrower the tracks and the smaller the spacing between the tracks, the finer the soldering tip you will have to use. This just makes it fiddly to solder if you are like me and have reached the age where your arms get too short 😊.

Once you have finalised your board layout and are ready to print, you need to ascertain as to how many PCB's you will require. If its just one board then it's a simply process. However, if you want multiple boards, you need to duplicate the boards on the layout software. In KiCad the PCBnew software allows you to select then duplicate the complete PCB. KiCad also allows you to have multiples of the same layout in the software, see the image below.

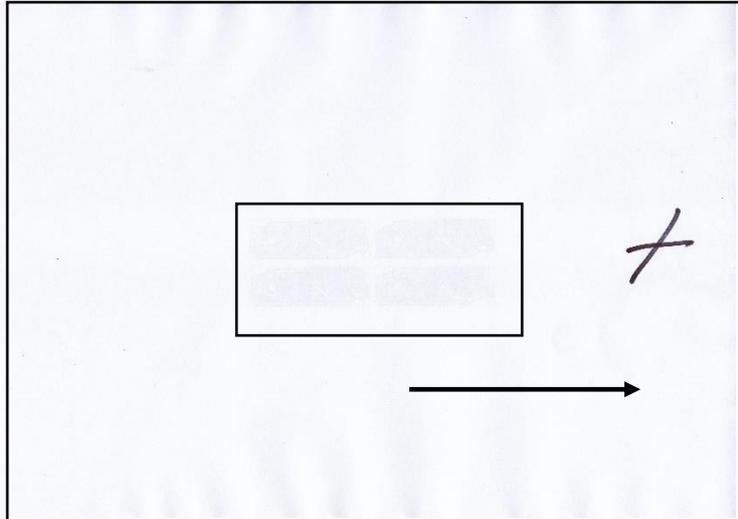


Before printing the layout on your laser printer make sure you have only selected the bottom layer and deselected all the other layers. As you can see from the above image, I have also orientated the PCB's to give me spacing in between for cutting purposes. The next steps are as follows:

1. Mark the paper in the laser printer tray. This gives me a reference to the orientation of the printed layout.
2. Do a print preview and make sure everything is correct.
3. Print a copy on the lowest dpi setting.

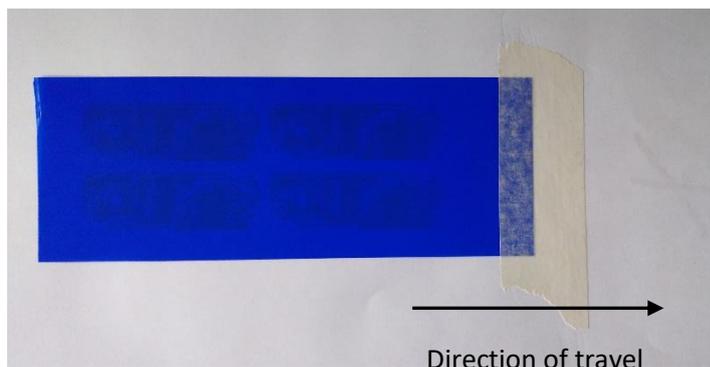


Print Preview

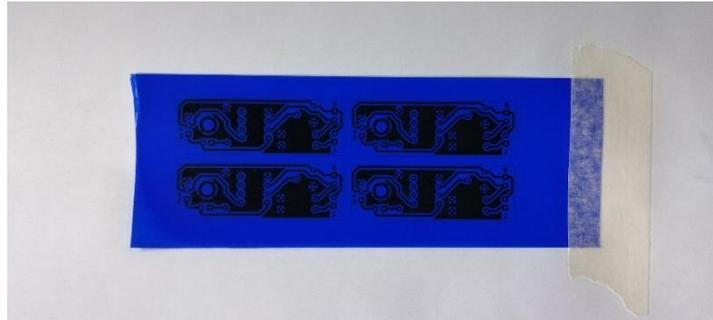


The X is my orientation mark and the arrow is the direction in which the paper travels through the laser printer. I have marked the centre with a rectangle as this is where the printed images of the PCB's are on the other side of the paper.

4. Now turn the paper over and place the blue transfer film dull side up over the image area. I generally cut the transfer film to a slightly bigger size than what is required.
5. Using masking tape, stick the transfer film down on the paper, taking note of the direction that the paper is going to travel in.
6. Place the paper back in the laser printer tray making sure the orientation of the paper is correct.

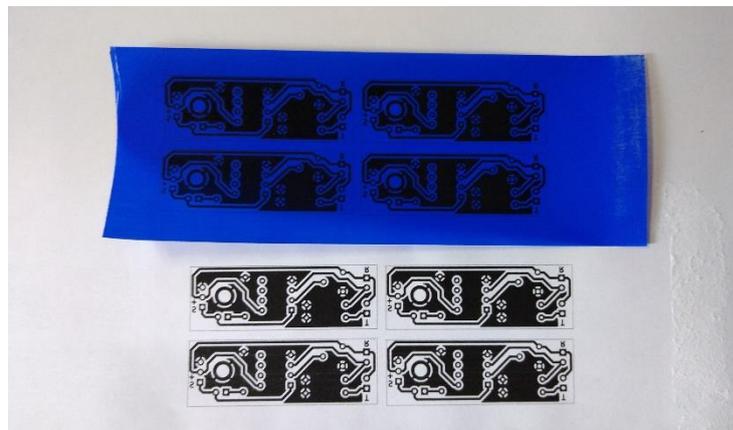


7. Before printing, go to the printer's preferences and reset the quality to 1200 DPI, and then proceed to print the layout. This is an important step for good quality PCB's.

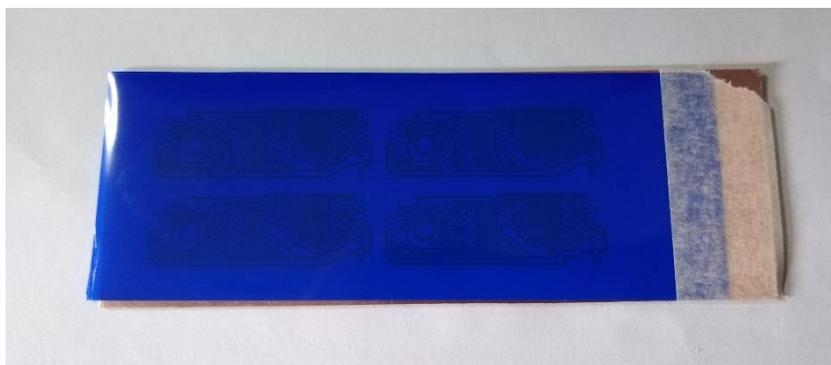


This is what your final print should look like when printed at 1200 DPI

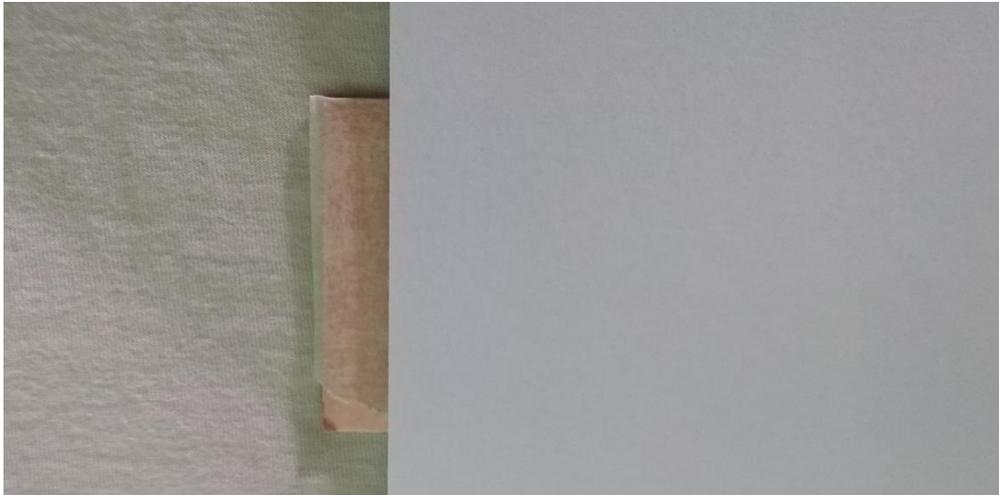
8. Remove the transfer film from the paper, see image below.



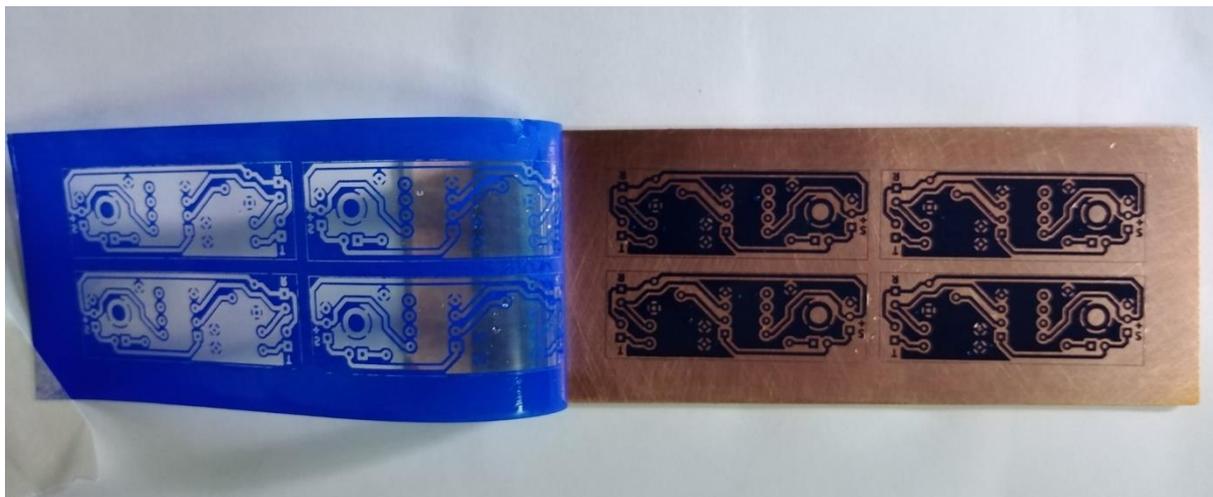
9. Cut out your PCB to slightly bigger than the transfer film.
10. Clean the PCB with the steel wool until shiny.
11. Now use the "Scotch Brite" or "Brillo" scouring pad with soap and water to clean the PCB once again. This is done to remove oil and any other contaminants from the surface.
12. Once the board has been cleaned, place the transfer film shiny side up on the PCB and use masking tape to hold it in place on the board. See the image below.



13. Now go and steal the XYL's iron and set it to the hottest setting. The instruction for the transfer film indicates a polyester setting. However, I have found this setting is too low. You might need to experiment to find the correct settings. But, from my experience the hottest setting should work.
14. Place the board on top of a soft surface such as an ironing board and place a sheet of clean paper, I use the 80gram paper from the laser printer, on top of the transfer film.
15. Start ironing the board from the masking tape side, don't let the XYL see you do it, as you might be roped into doing your own ironing from now on.
16. Press down on the iron to apply pressure. Iron for about 2 to 3 minutes. The PCB will get extremely hot and you will not be able to touch the board.
17. Lift the clear paper and check that the film has adhered to the PCB, if not then continue ironing the board until the film adheres to the PCB.
18. Let the board cool down for a minute and then rinse the board under cold water.
19. Once cool, you should easily be able to remove the transfer film from the board.



The PCB ready to be ironed.



The completed board after ironing

20. As you will see from the above image, I have peeled away the transfer film leaving behind the layout on the PCB. Note the water drops and the film, where the toner has been removed. This took around 3 to 4 minutes to complete from the start of the ironing process.
21. Check each individual PCB to make sure that the tracks are complete and that there is no copper showing through where tracks or the ground plane needs to be.
22. If there is a problem with the tracks or pads, use a fine black permanent marker to cover the area. These days you can buy 0.2mm permanent marker pens.
23. You can now use your Dremel or similar tool to cut out each individual PCB.
24. Once completed, place them in the solution of Ferric Chloride, until all the copper has been etched off the board.



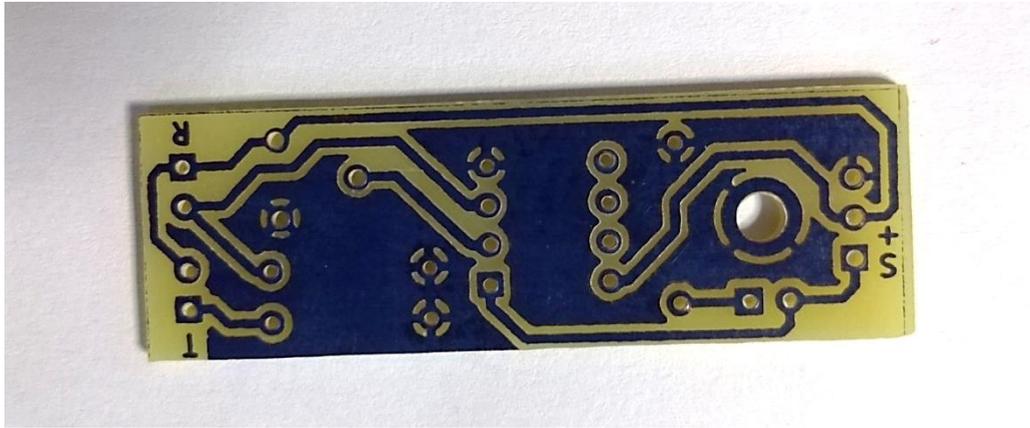
The PCB cut and ready for etching.



The PCB's after etching.

Now for the drilling and finishing touches:

25. Use a 0.8mm drill and drill all the holes. Take your time as accuracy is important.
26. For most components the 0.8mm drill will suffice. However, you would also need a 1mm drill for pin headers and even a 1.2mm drill for other components such as potentiometers etc.
27. The mounting holes are all 3mm



The final PCB

Leave the toner on the board until you decide to populate the PCB. This will protect the copper from tarnishing. To remove the toner, you can use Acetone or thinners. Before soldering make sure the copper side of the board is clean and free of all contaminants.

Take up the challenge and make your own PCB's for your next project.

73

Noel

ZR6DX