

# MAKING YOUR OWN PCB AT HOME (UPDATED)

By Bruce Pierson



*Above: Some materials for making your own PCB*

Anyone with an interest in electronics and the construction of projects, will from time to time, require a printed circuit board. Whether you are building a project from an electronics magazine, such as Silicon Chip ([www.siliconchip.com.au](http://www.siliconchip.com.au)) or whether you have designed your own circuit, the time comes when you need the printed circuit board to be able to assemble the project.

There are several options available to obtain the printed circuit board (PCB). Often, the project designer will have the PCB available for sale, or a specialist PCB manufacturer may have a range of PCBs available for various projects, or you may wish to produce your own PCB for a number of reasons, such as saving the cost of postage, or just a general interest in making your own PCB. You may also wish to alter the design of the original PCB to suit some special requirement that you have when constructing the project. In this case, you have to make your own PCB, as this is the only economical way to obtain the exact PCB that you require.

If you are going to make your own PCB, the first step is to obtain the layout of the board. Often, the designer of the circuit will have the layout available for downloading, unless the layout is propriety. In this case, making your own PCB becomes very difficult, but if the layout is available, then, with some basic equipment, you can make a reasonably professional looking PCB for your project. Of course, if you are designing your own PCB, then you already have the layout.

After having obtained the layout for the PCB, the first step is to transfer this layout onto a blank PCB, ready for etching. The most professional results will be obtained by using photosensitive PCB material and various developers and other chemicals. While this will produce a very high quality board, the cost can be equal to or in excess of purchasing the PCB from one of the available sources.

A more basic approach is to use a standard PCB blank and etch the PCB with ferric chloride or another suitable etchant. There are advantages and disadvantages of using different chemical etchants and each person should decide which etchant is the most suitable for them to use. My personal choice is ferric chloride, because this is readily available and I only make a small number of PCBs occasionally.

After conducting some research and adapting my findings to suit my needs, I have found what I believe is the most practical method of producing a reasonably good quality PCB at minimum cost with fairly basic equipment.

**The following materials and equipment will be required:**

Blank copper clad PCB material of the correct size.

Overhead Projector (OHP) transparency film and paper for printing the layout.

Access to a computer for organising the layout ready for printing.

Access to a LASERJET printer or a PHOTOCOPIER (NOT an INKJET PRINTER).

An electric iron for transferring the layout to the PCB.

Ferric Chloride or other etchant and suitable containers with lids.

Hobby drill and suitable (SMALL) drill bits, a hacksaw blade and a file.

**Safety Equipment:** Dust mask, eye protection, rubber gloves, tongs and newspaper.

The first step in the process is to obtain the PCB layout and print a reverse or mirror image onto a sheet of OHP film. OHP film is substantially cheaper than the specialised "Press 'n' Peel PCB film" available from electronics suppliers and works just as well. Often, the PCB layout is supplied as a normal image, but many printers have the option to print mirrored. If your printer doesn't, then it will be necessary to reverse or mirror image the layout. If the layout is supplied as an image file, then just open the image file with a graphics program such as MS Paint to flip the image and save the reverse or mirror image layout ready to print.

If the layout is supplied as a PDF file, then making a mirror image is somewhat more difficult. However, this can be done by first printing the layout onto paper, then scanning the printed page as 200 DPI line art and saving the scanned image as a 16 color or monochrome bitmap. Use your graphics program to flip the image and then save it. Once this is done, print the layout at 50% onto paper and compare the size with the original non-mirror image to make sure that the reverse or mirror image is the correct size. Some adjustment of the size may be required when printing. This can be done by selecting the "Adjust to [50 + or -]% normal size" in Page Setup Options.

Once you are happy with the size of the image, it can be printed onto a sheet of OHP film with a laserjet printer. If you only have an inkjet printer, then just print the layout onto paper and then go to your local library, newsagent or office supply company where you can gain access to a photocopier to print the layout onto your OHP film.

**Clarification on printing:** Inkjet printers use ink to print. This ink is not suitable for the next process which will transfer the layout onto the blank PCB. On the other hand, laserjet printers and photocopiers use a plastic material (the black powder) which is actually melted onto the paper or OHP film by the printer or photocopier.

Once you have your reverse or mirror image layout printed on the OHP film, it is time to prepare the blank PCB for transferring the layout onto it. The first step is to cut the PCB to the correct size. This can be done using a fine tooth metal cutting hacksaw blade. The cut edges can then be filed smooth. Be careful not to cut yourself on the sharp edges after cutting the PCB. **Note:** Use a dust mask when cutting and filing the PCB and vacuum up all dust when finished. (Refer to drilling instructions).

The next step is to clean the copper surface of all oxide, fingerprints, grease, dirt and other contaminants. A variety of methods can be used. I have found that 1200 wet and dry paper and liquid soap with water produces a good result. Lightly rub the board in a circular motion until the copper is shiny clean. A non-metallic scourer can also be used. Do not use normal steel wool as this can lead to iron contamination of the copper. However, stainless steel wool can be used as long as it is not too coarse. Rinse and dry the board thoroughly after cleaning.

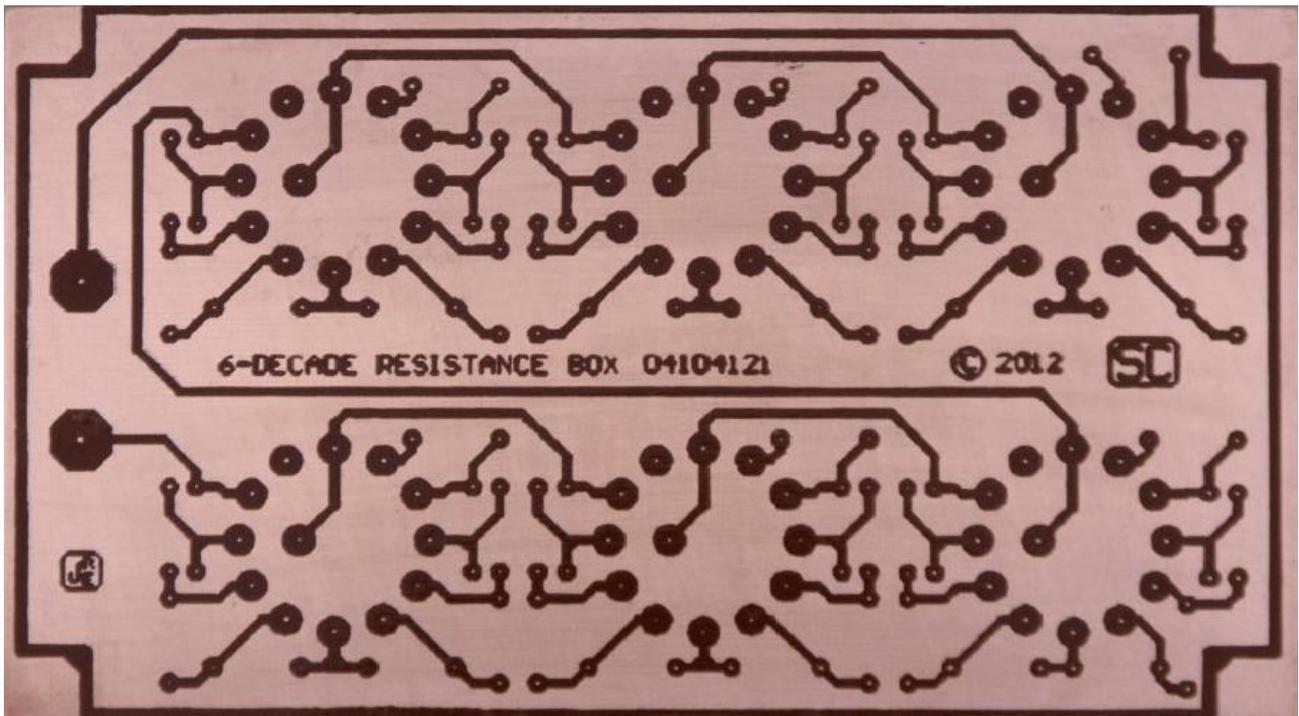
Now it comes time to transfer the layout onto the PCB. Firstly, pre-heat your iron to the hottest temperature it has. Work on a surface that will not be affected by heat, such as a block of wood or a flat concrete floor. Place the blank PCB copper side up on the work surface and carefully lay the OHP film over the PCB with the layout lined up correctly with the PCB and the printed side in contact with the PCB.

Hold the film in place and put a single sheet of paper on top of the film. Put your hot iron on top of the paper and press firmly, but not too heavily. Move the iron around in order to heat the entire PCB evenly. Continue this heat and light pressure for several minutes in order to ensure that the layout is correctly transferred to the PCB. When you are happy with the heating process, allow everything to cool **completely** before proceeding to the next step.

Once the PCB is totally cold, lift the OHP film and it will very easily peel off, leaving the layout on the PCB. If sufficient heat was not applied during the last step, it may be noticed that the layout has not transferred completely and the result is not suitable for etching. All is not lost. Print another layout onto a new piece of OHP film and repeat the previous process, being extremely careful to exactly line up the new layout with the existing partial layout on the PCB. If the failed attempt is so bad that it is considered a total write-off, then firstly clean off the failed layout with a suitable solvent such as general purpose thinner or acrylic lacquer thinner and start over.

Note that minor defects can be touched up using a Dalo Pen or a permanent marker such as a Pilot Marker. Not all types of permanent marker may be etchant resistant, so some experimentation may be required to ascertain the durability of a chosen type of permanent marker. Note that water based markers are not suitable, as the ink will simply dissolve when you start etching the PCB.

The PCB featured here is Copyright © Silicon Chip 2012. The PDF layout can be downloaded from the Silicon Chip web site ([www.siliconchip.com.au](http://www.siliconchip.com.au)).



Above is a PCB which has been prepared for etching, using the method described.

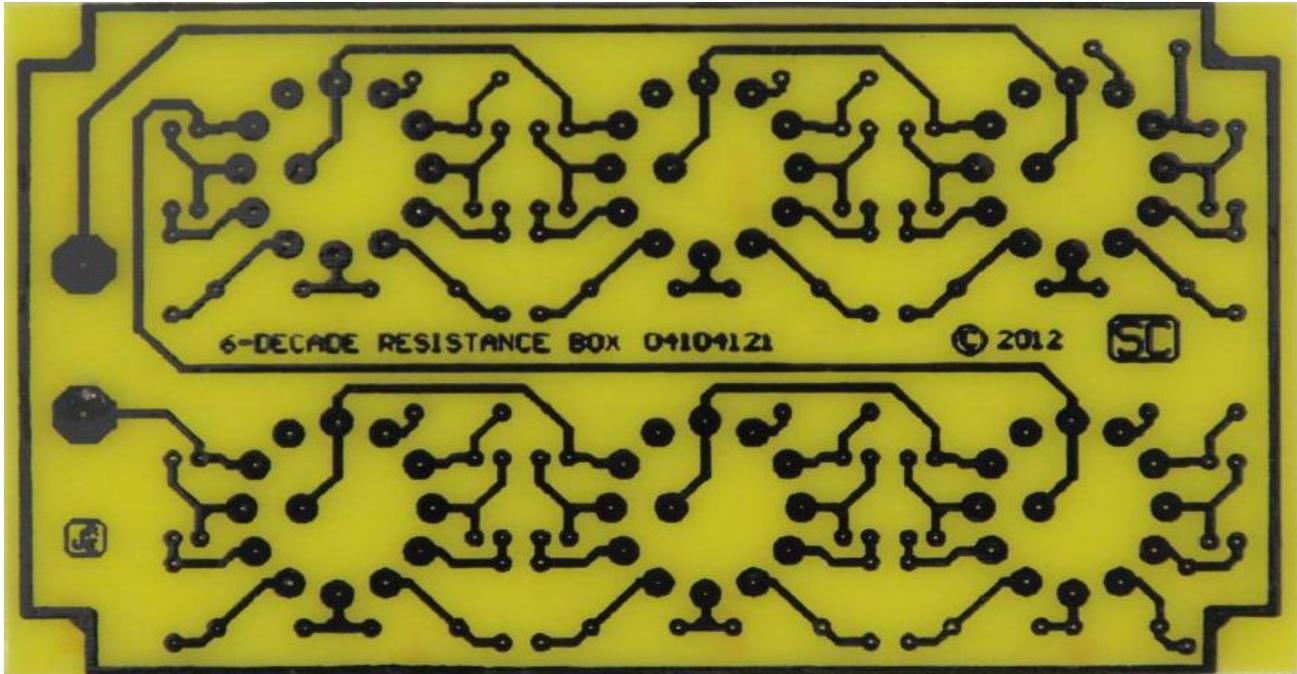
Once you are satisfied that the PCB is ready for etching, the next step is to prepare the etchant. The instructions will be for Ferric Chloride, so if you are using a different etchant, you should research the best method for that particular etchant.

Work in an open area, preferably outdoors and on a surface that will not be affected if any ferric chloride is spilled. Several sheets of newspaper can be used to cover the work area in case of spills. Be aware that ferric chloride is corrosive and will badly stain clothing and other surfaces, so avoid contact with it. Wear rubber gloves to protect your hands and use tongs to insert and remove the PCB. The use of safety glasses or a face shield is recommended to prevent eye contact in case of splashes.

Obtain a plastic container that is a little larger than the PCB to be etched and pour sufficient ferric chloride into the container to a depth sufficient to cover the PCB when immersed. Place this container into a larger container with sufficient room to easily accommodate the container with the ferric chloride. Into this larger container, add very hot to boiling water. This will heat the ferric chloride solution and greatly reduce the etching time. Be absolutely certain that your containers are capable of containing very hot to boiling water, to avoid failure of the containers and the possibility of severe scalding (burns) from the hot to boiling water.

After a few minutes, when the ferric chloride has heated, carefully immerse the PCB in the ferric chloride, copper side up and rock the container from side to side, being careful not to spill the ferric chloride or the hot water. More hot water may be added to the larger container after a few minutes to keep the ferric chloride heated. Etching should take around 10 - 15 minutes, depending on the temperature and strength of the ferric chloride.

Keep a close eye on the etching process and once the excess copper has been removed, take the PCB out of the ferric chloride and rinse it in clean water and wipe it dry. The ferric chloride can be stored in the container used for etching. Ensure that the lid is replaced securely and store in a safe place, well out of the reach of children and others. Place a label on the container and lid to identify the ferric chloride.



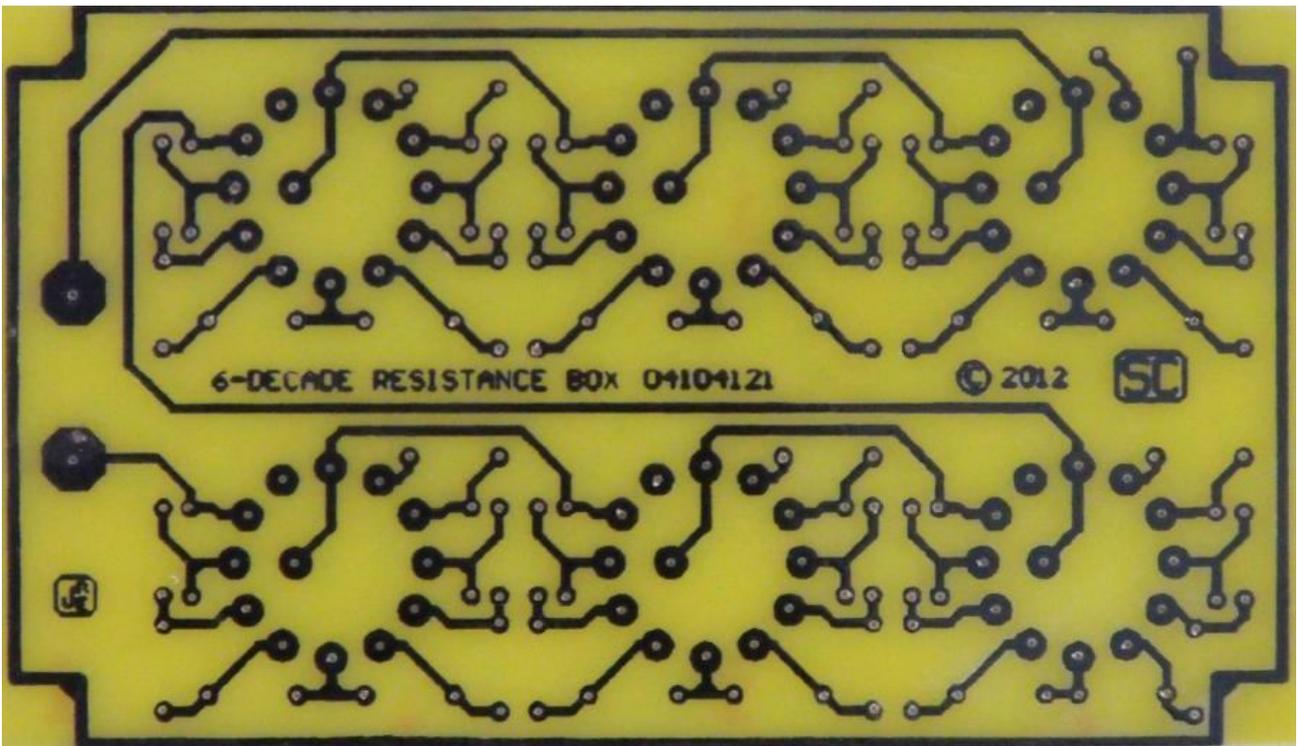
*Above is the PCB which has been etched using the method described. It is ready for drilling.*

The next step in the process is to drill the PCB ready for insertion of the components. If you are using surface mount components (SMD) this step will not be necessary, other than holes for wires, etc.

A small hobby drill is suitable for drilling the board. Work on a piece of scrap wood, so that the work table will not be damaged by the penetration of the drill through the PCB. The etching process should have removed the copper from the areas to be drilled. Some small amount of copper may be present in some of the areas to be drilled, but should not present a problem. Carefully drill each hole, being sure to centre the drill in the exact position for the hole.

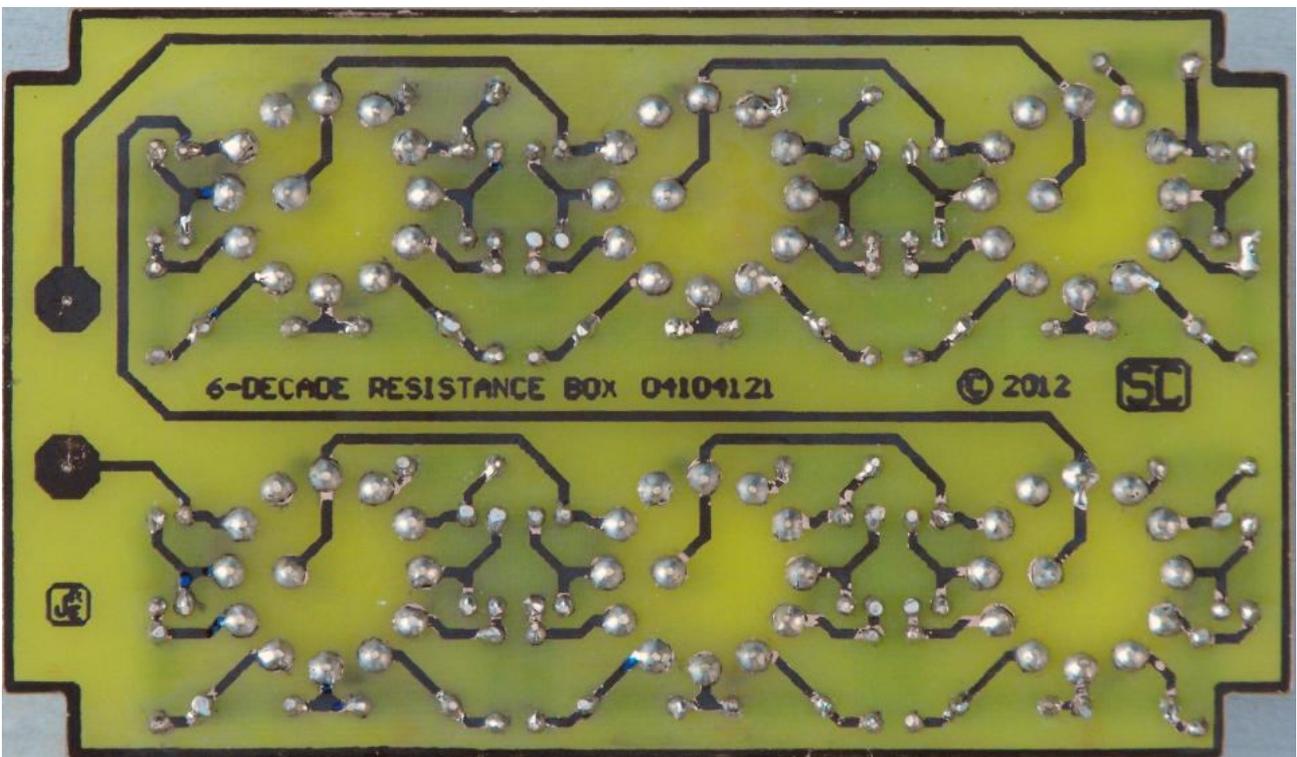
It is important to drill the correct sized holes for the components to be used. Typically, most holes will be 0.8mm for smaller components, 1.2mm for larger diodes and high wattage resistors and around 2mm or more for larger components with thicker leads and thicker wires. It is most practical to drill all the holes 0.8mm first and then enlarge any holes for larger components and wires as necessary. Check your component layout for any holes that need to be larger and drill to the correct size.

**Note:** It is recommended to wear a dust mask during the drilling process and to thoroughly vacuum up all dust when finished. Modern PCBs are usually made from fibreglass and it is not desirable to inhale this dust as it may be detrimental to your health.



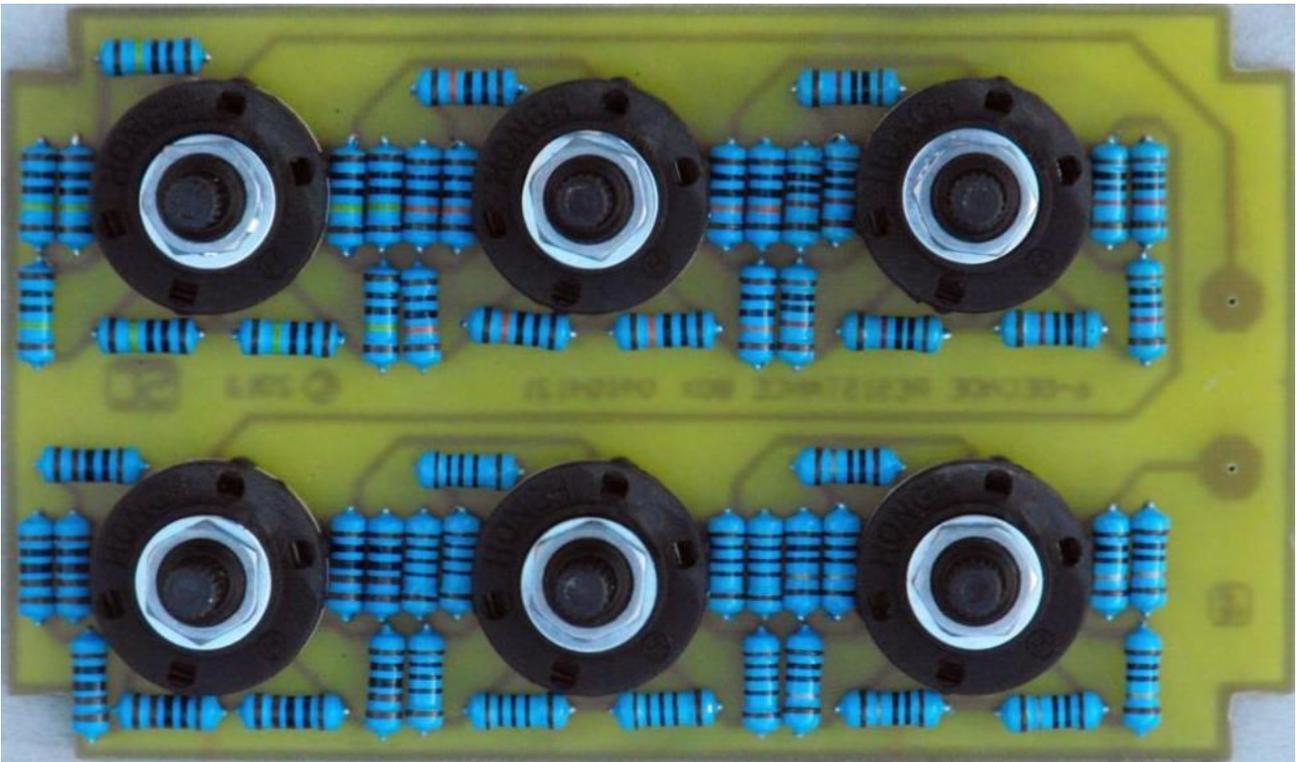
*Above is the PCB which has been drilled ready for component assembly.*

This completes the process of making your own PCB and you can now continue with the fitting of components and wiring for the project construction. There is no need to remove the printed layout, as it can be soldered through.



*Above is the PCB which has been trimmed, assembled and cleaned.*

After soldering the components to the PCB, the PCB can be cleaned by scraping the resin residue off the PCB with a suitable tool, such as a small screwdriver, being careful not to damage the PCB. The board is then washed with a small quantity of methylated spirits and an old toothbrush, before being dried with a soft cloth.



*Above is the component side of the PCB after assembly. The original project used  $\frac{1}{4}W$  resistors, but in this case 1W resistors have been used instead.*

If required, the same process can be used to print a component overlay on the top of the PCB after drilling the PCB. A coating of clear automotive acrylic lacquer can be applied over the component overlay to protect it from damage.

## **Troubleshooting and FAQ**

### **FAQ**

**Q.** I followed the steps, but when I went to remove the OHP film, I could see that the layout had not transferred properly to the PCB, because a lot of it was still on the film. What caused this and can I put the film back down and apply the iron again to fix this or will I have to start again?

**A.** This is caused by not enough heat during the ironing process. If you have only partly lifted the film, you can put it back down again and re-apply the iron. If you have taken the film off completely, you may be able to re-use it if you can carefully line it up again. Otherwise, just print a new layout and carefully line the new layout up with the existing track-work, or clean the board and start again.

**Q.** After I removed the film, the tracks were all blurry and smudged and running into each other with no clearance between the tracks. What caused this and what can I do.

**A.** This is caused by either too much heat and pressure from the iron or the film moving while ironing. If only a small section of the layout is affected, it might be possible to carefully scrape the side of the blurred track off with a sharp knife, such as a Stanley Knife (box cutter) to provide sufficient clearance to the adjacent track. If the area affected is extensive, it is better to clean the board and start over again.

**Q.** I have made a PCB. Can I re-use the OHP film to make another PCB?

**A.** If you have only printed on one end of the OHP film, you might be able to use the other end, depending on the condition of the OHP film. Usually, the OHP film is damaged and distorted during the ironing process and often small pieces of toner are left on the film. If the film is not distorted, it can be cleaned and re-used (print the other end) or the damaged end cut off so that the remaining piece can be used for the next PCB. **DO NOT USE DAMAGED FILM IN YOUR PRINTER AS THE PRINTER DRUM MAY BE DAMAGED.** A new piece of film is much cheaper than a new drum. I prefer to cut the OHP film to size before ironing it, so the resulting piece of used OHP film is too small to reuse anyway.

**Q.** I want to make a PCB and I want to print a component overlay on the top of the board as well. Is there any special steps I should take when doing this.

**A.** The best procedure to follow is to first print and etch the PCB, then drill the holes, then wash and dry the PCB thoroughly. The component overlay can then be lined up accurately with the holes in the PCB and the same procedure followed as for transferring the track-work. If you have left the printed track-work on the solder side of the PCB, then place the PCB onto an off-cut of OHP film while ironing, to prevent the printing from being damaged or possibly partly transferring to the work surface. After successfully transferring the component overlay and removing the OHP film, the off-cut of OHP film on the track side will easily peel off again, as the heat transfer to the track side is not as great as when transferring the track-work.