

Printed Circuit Board Production using Computer Aided Design and Photo-Resist Technique

E.C. Arihilam, Eguzo C.V. and Osazuwa E.E.

*Electrical/Electronics Department
Akanu Ibiam Federal Polytechnic Unwana, Ebonyi State, Nigeria
E-mail: engrarihilam@yahoo.com, chimaxcorporations@gmail.com,
efoski@hotmail.com*

Abstract

Producing a Printed Circuit Board layout can be a time consuming exercise and the final art work needs to be accurately produced if success is to be achieved. Using the computer as tool in the production of PCB layout overcomes any of the problems and is a valuable computer application. The Express PCB and the Express Schematic software allow the construction of an accurate layout, from which a hard copy can then be produced on a printer. This is then processed with the photo-resist technique, etched, drilled and populated by components. From the current flow through the various track sizes produced, we found out the more the quantity of current flow, the more the track width.

Keywords: Photo-sensitive, Photo-resist, Express SCH, Express PCB, Ultraviolet, Etch.

Introduction

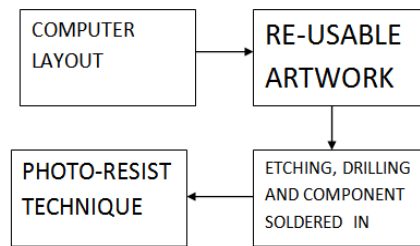
Producing a printed circuit board photographically (i.e using the Ultra Violet or Photo-resist technique) only differs from the simple method of board production in the way in which the resist is applied to the board and the required pattern is obtained. While it must be admitted that this system of board production is more expensive, it does not allow really good result to be obtained.

An advantage of this method of board product is that modern, highly intricate printed circuit board can be produced without too much difficulty. Again, it is very quick and simple to produce a number of identical boards using this system, and this is useful if you wish to build two or three examples of a project for some reason or where a project uses several identical circuit boards. This system is also very useful if

you are designing and building your own printed circuit boards since you can draw the printed circuit track design on translucent drafting film and then this actual size drawing can be used as the photographic positive to produce the board.

Production Route

The production route for the PCB using computer aided design is given below;



Computer Layout

These days there are softwares for producing layout and final artworks from the production of PCB. As mentioned earlier, designing a PCB layout artwork can be very time consuming but with good software, life can be made extremely comfortable.

The software used for the computer aided design is a demo version of express PCB and Express Schematic. They are easy to use windows application for drawing schematics and PCB

Re-Usable Artwork

The term artwork applied to a design which shows the tracks, pads and the component position desired for a PCB. Being able to re-use this art work is great advantage. The hard work of laying out the circuit will not be wasted as the master can be stored for later use or adaptations. Transfers can be used for this purpose as they are accurate in dimensions and fairly easy to lay. They also help with the spacing of pads and give stable transparent film and the work is created to look like the final PCB when viewed from underneath.

The Photo-Resist Technique

Using a photo-resist board is one of the easiest ways of producing a PCB, the photo-resist board used is coated with a substance which is sensitive to ultra-violet light. After exposure, the board is submerged in a developer which is a solution of dilute sodium hydroxide solution. The area that the mask protected from the UV light will now become protected for the etching process. When the board is in developing solution, it will gradually turn pink. The area that was masked should slowly become defined by a dye in the photo sensitive material, this is an indication that the process is working.

Etching, Drilling and Component Soldered in Place

The etching process will remove all unwanted copper, leaving the tracks and pads desired. There are many possible etching solutions but dilute ferric chloride is the most popular. To be sure of success the solution must be warmed to around 40 degrees Celsius; if the solution is too cold it will be difficult to achieve a successful etch.

A drill that is meant specifically for PCB job has a stand that is particularly important for accurate work and will help to extend the life of drill bits. For general PCB work, the most suitable drill has a 0.8mm diameter. Most component pins or wires will fit this size and ICs in particular are easy to locate. The 1mm diameter drill is more substantial and a reasonable alternative. The important point to remember is that the closer the wire fits in the hole, the easier soldering will be. Both an electric soldering iron and multi-core solder are essential but the important factor is to ensure a good, clean joint.

Materials and Methods

Materials

The Artwork Light Box

The artwork light box gives an even illumination of the A3 size opaque acrylic work surface. This is achieved with two 15W fluorescent tubes contained in a rigid steel or wooden case. The light box is usually used with a precision grid and aids the production of accurate artwork. It is recommended that any area of the work surface not being used is masked with thick paper or card; this will protect the eyes from any undue tiredness caused by long spells of concentration.

Transfer

Transfer symbols are mounted on a 25 micro transfer film which is very stable, and resist cracking and deformation. This film is particularly appropriate where accurate work is required and it has a track width range of 0.38mm, 1.02mm and 2.03 mm.

Developing Chemicals

A 7 grams of caustic soda to one liter of water forms/makes a standard caustic soda developer solution (sodium hydroxide solution). It is essential to make up a solution of the right strengths since the developer only needs to be slightly weak in order to almost totally block development. If the solution is slightly over-strength, it is likely that the unexposed areas of photo-resist will tend to dissolve almost as rapidly as the exposed areas, resulting in an inadequate wasting of resist or no resist at all after development.

UV Exposure Unit

This unit is designed to give an even exposure of ultra violet light to photo-sensitive board. A timer is incorporated in the unit which allows accurate control of the exposure. The larger unit has a bigger working surface of 330x260mm while the

smaller unit has 229x150mm and four 15w UV tubes contrasting with the two 8w tubes of the economy unit. The UV radiation from the tubes is not of a wavelength that could cause eye damage when in normal use; however, it is good practice to avoid unnecessary exposure.

Photo-Resist Board

The board is epoxy glass and is 1/16" thick with a 35 micro copper coating which is itself covered with a thin layer of photo resist. The photo resist is protected by an opaque plastic film. The film is removed immediately prior to exposure in the UV unit. It is possible to extend the shelf life of the boards by storing them in a refrigerator, otherwise it is advisable to use the board with a few months of purchase.

Bubble Etcher

The bubble etcher is designed to contain a corrosive etchant at the correct working temperature. It agitates its solution so that a good consistent etch is produced. To achieve this a 300w thermostatically controlled heater element is fitted and an internal pump forces air through bubble bars. Once the correct temperature for the solution is reached the solution is ready for use. It is advisable to mix less than 5 liters (the capacity of the tank) to minimize spillage. Because of the need for safety when handling ferric chloride solution, a syption is included which allows a more controlled ways of decanting the solution.

PC Board Drill

A rigid drill stand supports the drill unit. The drill is controlled by a variable 12V supply and appropriate speeds can be set depending on whether you are using the conventional high speed steel drill or the harder tungsten carbide drill bits. The drill chuck can be locked with the spindle lock button and drills can be tightened in a place by hand. The chuck has a capacity of 0.4 to 6mm. The maximum speed of the drill is approximately 18,000 r.p.m. The drill has substantial bearings but undue pressure should be avoided to preserve the life of the unit.

Soldering Iron

An 18w soldering iron with a small tip is suitable. It is safer to use an iron powered from a low voltage supply, one of the dangers when soldering is that a carelessly placed hot iron can burn through the flex. For this reason and generally convenience it is worth using a stand which often includes a sponge that can be damped to clean the soldering iron tip.

Methods

Computer Aided Design Layout

Using the computer as a tool in the production of PCB layout overcomes many of the problems in producing a final artwork and is a valuable computer application. The Express PCB and Express Schematic software allows the construction of an accurate artwork, from which a hard copy can then be produced on a printer or plotter.

This hard copy (layout) forms the component (upper) side while the artwork is on the copper side of the board. And to have a re-usable artwork, all you need to do is to make a good photocopy of the artwork and then ask a lithographer to produce 1:1 positive on a film. The finished artwork with this method has the advantage that it is more difficult to scratch the tracks and pads unlike the finished artwork using the “transfer method”.

Development

The following steps can be followed;

1. Make up a dilute solution of sodium hydroxide made of 12g of sodium hydroxide mixed with one liter of warm water (20⁰C) this solution should be handled with care.
2. Switch on the UV exposure unit and allow it to warm up for a few minutes, switch off before opening the unit to lay the mask.
3. Check that both the mask and glass are clear of any dirt which could affect the result.
4. The photo-sensitive board protected by a film of opaque plastic must be peeled off before the board is used. The exposure surface is sensitive to the UV part of the light spectrum. Ideally the opaque plastic film should only be peeled when the board is ready to be exposed.
5. Place the board over the mask, sensitive face down
6. Close the lid, set this time and switch the UV light. An exposure time of approximately four minutes is recommended but this time may vary.
7. Once the board has been exposed it should be possible to make out a vague outline of the track. Carefully remove the board and using tongs, place it in the sodium hydroxide solution.
8. Agitate the solution gently to ensure that the developer works evenly. At first the board will be dark purple but as the solution begins to work the tracks will become well defined, the photo-resist will lighten in colour and it should be clearly seen dissolving in the solution, continue this process until the track is well defined in purple and the copper background is fully visible.

Etching

1. Place the developed and washed PCB into the bubble tank containing ferric chloric solution
2. Carefully remove the PCB after approximately eight minutes and visually check the process. Generally, etching will take about twelve minutes depending on the quality of the solution but is worth checking the progress of the board occasionally.
3. Once the copper track is well defined, remove the board with the tong and again wash it thoroughly.
4. Check the PCB for any flaws. Small cracks in the tracking will obviously spoil the circuit.

Drilling

1. When drilling only apply gentle pressure to the lever and be very careful of any bending of the bit itself. The drills particularly tungsten carbide are very fragile
2. Once all the holes are drilled the PCB should be examined closely. The drill would have produced burrs which should be lightly removed with an abrasive.

Design

Using the CAD System

This is an illustration of a procedural approach towards using the CAD system for the prototype electronic buzzer circuit diagram shown below

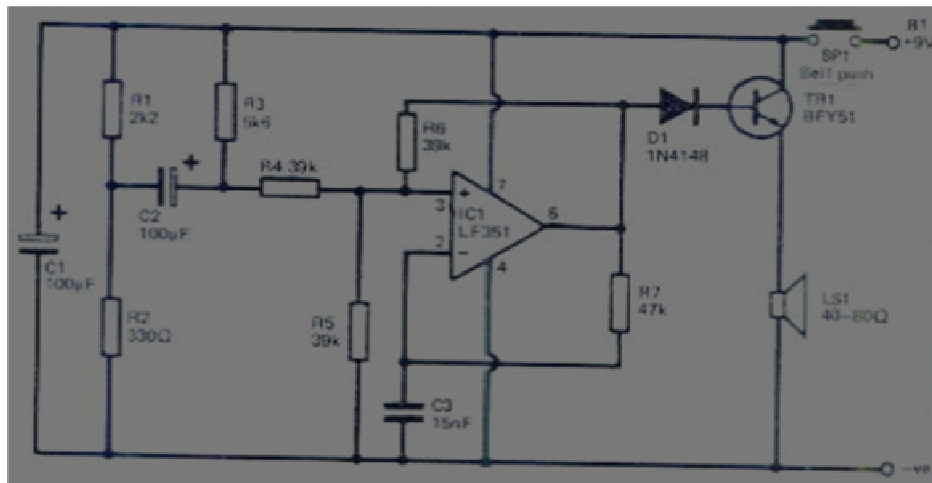


Figure 1.1: Electronic Buzzer Circuit.

Step One: Drawing the Schematic

In Fig 1.2 below is the basic layout view of the Express SCH showing most active component of the circuit (the IC OP-AMP 351) selected from the component and symbol manager tools box. In the same way all the other components will be placed on the drawing layout of the software before establishing their electrical connections.

After a perfect placement of all the components and establishing good electrical connection between them, the schematic diagram produced is that shown in fig 1.3 below.

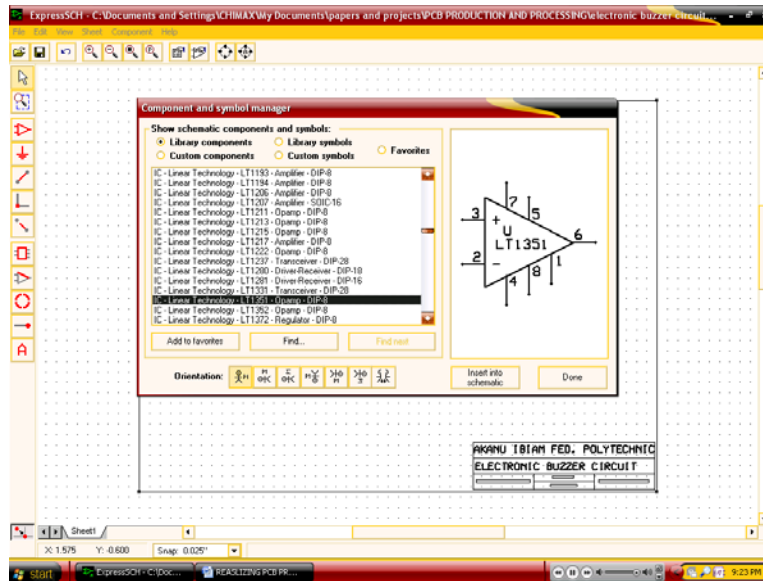


Figure 1.2: Component placement using component and symbol manager.

Step Two

Another important step will be to name a part identification for all the components to differentiate say a resistor from the other. This is very important when tracing and drawing the PCB. To name a part ID, double click the component to open the component properties window, then change the part ID.

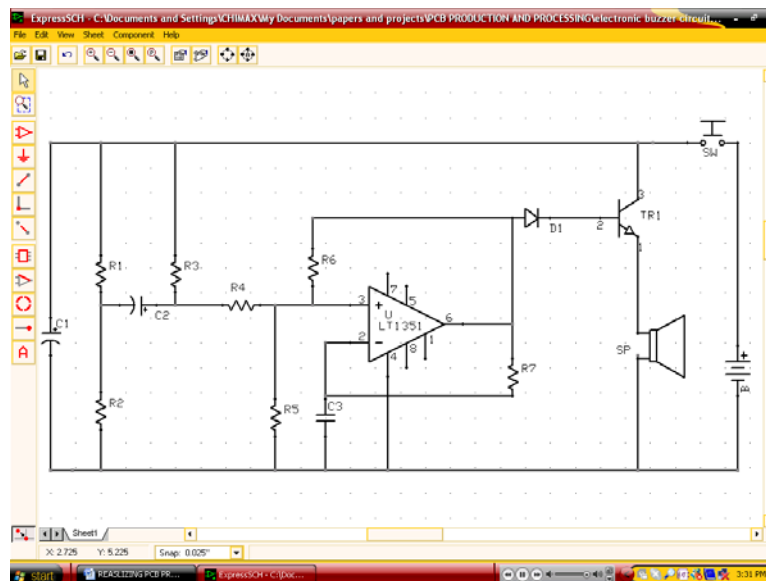


Figure 1.3: Completed schematic diagram of Electronic Buzzer showing the part IDs.

Step Three

The last step of this section is to check the schematics for netlist errors from file menu after which you save and close Express SCH.

Step Four: Transferring Schematic to PCB

In this session, the Express PCB software will be used to draw the component layout, the circuit artwork and link the already existing schematic from Express SCH to Express PCB in completion. Components placement require accurate use of the grid lines to allow for board drilling and literal transfer of physical components to a completed artwork using standard grid spacing. The spacing permissible for IC (e.g 8 pin DIP) is 0.1 inches hence set the grid spacing to 0.1inches from the options tools box. Also choose your component from the component and tools manager. At the end of the component placement, the produced component layout is as such produced in fig 1.4. It is worthy of note here that off board components like power supply and switches are not found in the components list for PCB but can be found in component list for schematics, in such case pad spaces will have to provided for their placement. A good component layout will highly reduce the use of jumper wire to complete electrical connections. This therefore calls for a careful placement of the components that will allow for symmetrical order and jumper free connection. Name a part ID for each of the components placed according to their different representations in the Express SCH software, then click on “link schematic to PCB” from file menu. To draw the PCB, click on “highlight net connections” from the tools box to show the components that electrically linked using the schematic as guide.

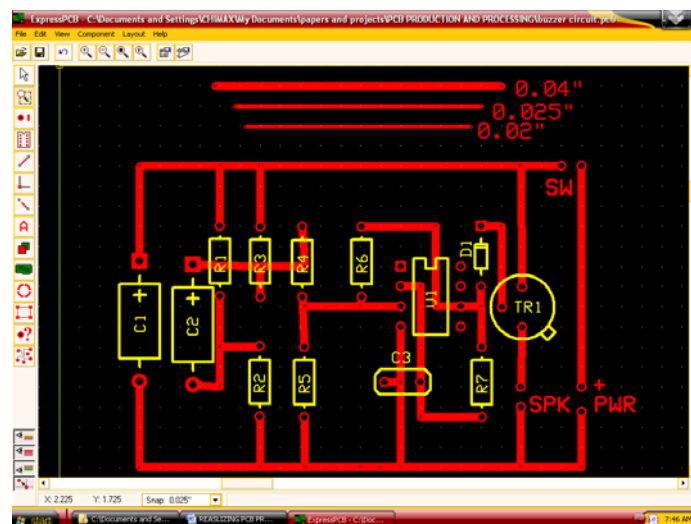


Figure 1.4: Printed Circuit Artwork and Component Layout View showing Traces of different sizes.

For the purpose of this paper, the chosen trace size is 0.04” for clearer view and understanding. Smaller sizes reduce the likeliness of trace crossovers.

Step Five

The last step will be to view and Print only the artwork without showing the component layout. To do this, click the option menu in the tool box and check the “view silk layer option”. The completed artwork is shown in fig 1.5.

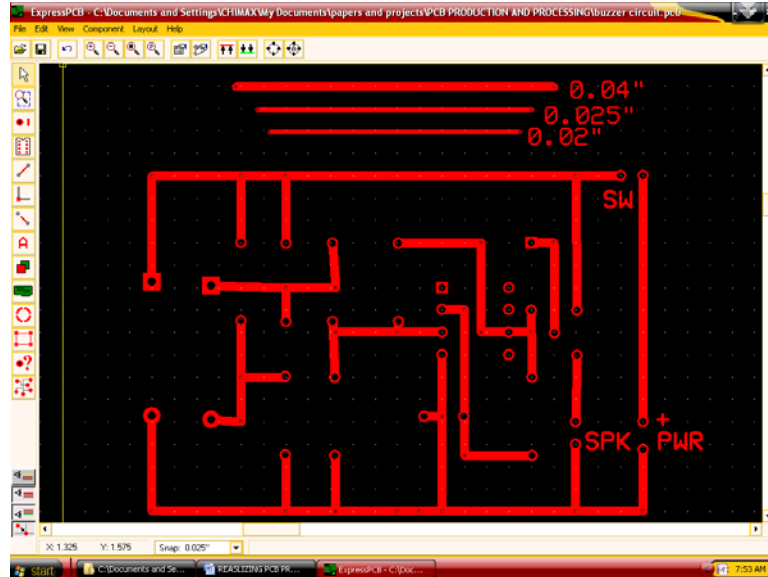


Figure 1.5: A completed artwork of buzzer circuit showing the copper traces.

Step Six

Finally print the artwork for onward transfer to the photo-sensitive board using the Ultraviolet Method. When using a laser printer, set the DPI to 600 minimum, output color black only and make it as deep as can be allowed.

Result and Discussion

For the various track widths on the board at 10°C , the following current flows were recorded using simple digital meter when the circuit became populated.

Track Width	Current Flow
0.1mm	0.45A
0.5mm	0.85A
1.5mm	2.95A
2.0mm	3.15A
2.5mm	4.50A
3.5mm	6.50A

As a rule of thumb, the larger the current required to be carried by the PC tracks, the wider the tracks must be. It is advisable that we go for a minimum width of 0.8mm. It is however worth finding the maximum current to be carried by the power rails and any track which leads to power devices such as relays etc.

Pad should be as large as possible as they are easy to solder. With Pad holes of 1mm, the pad diameter should be 3mm. Shaped pads for ICs obey this general rule in one direction but not between pads to allow the possibility of laying tracks between the pads.

Conclusion

From the afore-going therefore, it can be established that The more the track sizes especially towards the power devices, the more the current flow in that section.

References

- [1] Arihilam, E.C (2007). Production of Electronic Printed Circuit board in schools, *Journal of Science and Engineering Development J.Sc and Engr. Dev.*,(Vol 1 No 2, 2007, pp 2 – 7)
- [2] Bonam Electronic UK Inc, (2009). A 3-Day practical training manual on PCB production/processing organized by the Nigeria society of Engineers and NIEEE Kaduna Chapter. August 2010.
- [3] ExpressPCB User's Manual (2009). www.expresspcb.com
- [4] George M. Chute and Robert D. Chute (1996), *Electronics in Industry*. MC Graw-Hill Book Comp, New York.
- [5] John D. Lenk (1995).*Handbook of simplified solid state circuit, Design*. Prentice Hall Inc, New Jersey, US.
- [6] R. A. Penfold (2003), *How to design and make you own PCBs*, London. Bernard Babani Publishing Ltd.
- [7] R. A Penfold (2007). *Electronic Project Building for Engineers*, London, Bernard Babani Publishing Ltd.
- [8] Roger W. Prewith (1986), *Electronic Device and Circuit Application*. Reston Publication Comp Inc. Yorkshire. A prentice Hall Co.
- [9] Testbed Tech (1987), *PCB testbed Test Blackburn*. BB 9TD, England.