PCB DESIGN USING LOCAL TECHNOLOGY AND AutoCAD

Orosun R. O1,3, Orosun M. M2, Salawu N.B4 and Ige S.O2

1Electrical Section, Factory Department, Savannah Sugar Company Limited, Numan, Nigeria.
2Department of Physics, University of Ilorin, Ilorin, Nigeria.
3Department of Electrical and Electronics Engineering, Bayero University, Kano, Nigeria
4BS Geophysical and Consultancy Ltd., Kwara state, Nigeria

Email: raphealorosun@gmail.com, muyiwaorosun@yahoo.com

Abstract. This paper focuses on the production of printed circuit boards for use with both domestic and industrial electronics. The use of printed circuit board (PCB) design software, methods of design transfer to copper boards, etching and post-production tasks were undertaken and are described in detail in this paper. The relevance of this paper is appreciated for its unique ideas and methods of incorporating local technology into modern PCB design with proven reliability, precision and economic advantages.

1. INTRODUCTION

The world today has people constantly looking for new ways of doing their conventional tasks with cost reduction and increased efficiency in focus. Today, various PCB design software are on sale to take care of both circuit diagrams and printed circuit boards designs. Some of them are Express PCB, Multisim and EAGLE. Ease of implementation, neatness, reduced production periods have been identified as advantages of using PCB over other methods of producing electronics circuits. We used EAGLE to produce the PCB for this project. But the knowledge can easily be adapted for any PCB design software, since the basics are similar irrespective of the software.

2. METHODOLOGY

EAGLE is a powerful graphics editor for designing PC-board layouts and schematics. The system requirements and technical specification of the software is listed below (CadSoft Inc., 2008). In order to run EAGLE the following hardware is required:
- IBM-compatible computer (586 and above) with
- Windows 95/98/ME, Windows NT4/2000/XP or
- Linux based on kernel 2.x, libc6 and X11 with a minimum color depth of 8 bpp,
- a hard disk with a minimum of 50 Mbyte free memory,
- a minimum graphics resolution of 1024 x 768 pixels, and
- preferably a 3-button mouse.
2.1. Programs user interface

Some screenshots of the programs user interface are shown below. When the program is started, it displays the control panel, showing the libraries, projects, design rules and other parameters used to achieve its functionality [1]. The program like any other schematic editor has the symbols for various electronics components built in and the user just needs to pick the required one(s) and place it on the board to get the drawing done. The toolbar to the left (see Fig. 1) contains tools for drawing schematics and arranging them appropriately.

Fig. 1. EAGLE Control Panel
Fig. 2. A typical Schematic sheet

The drawing is done by picking and placing components and then using the NET or WIRE tool to interconnect them appropriately. A well drawn diagram is shown in Fig. 2 above. After the drawing is completed, the BOARD command (used for creating the board from the schematic) is invoked and the board opens up in another program window showing the life size components and their interconnections. It is illustrated below:
Fig. 3. Board showing life-sized components and their interconnections

The actual board is the rectangle to the right whose size can be re-adjusted to meet the designer’s specification. At this point, the components can be arranged as the designer deems fit on the board having considered factors like air flow, isolating temperature sensitive devices from heat generating ones etc. At first sight, the user may wonder how to deal with so many lines, but the program binds each component to its connections as defined in the schematic irrespective of its location on the board. After this has been done, the autorouter, on the toolbar to the left as seen is invoked. The autorouter is responsible for generating the PCB layout using a rip-up and retry algorithm to achieve the desired design. Before actual routing is begun, the user defines the routing parameters, like specifying whether the board is a double-sided board or a single sided one, the preferred routing directions and other settings [2, 3].

A screenshot is captured below (see Fig. 4.):
Fig. 4. Routing Setup dialog box

After routing is complete, the board is left with tracks defining the circuit connections and then the components can be shown or hidden. A finished routing job is shown below without the components. The routed circuit is that of an Automatic Car Security System and is a single-sided board [1, 3, 4].
Fig. 5. A complete Routing Job.
As can be seen from the drawing, the blue lines indicate the proper connections while the green-colored sections are the solder pads with which the components are mounted on the board. After this stage the output is now transferred to the board, however, there are various methods of achieving this transfer and they are outlined below.

3. TRANSFER METHODS

There are about four different methods for transferring the output of PCB design software to a copper clad board. They are listed below.

- Mesh and Ink Method
- Direct transfer
- ‘Press n Peel’ sheet / Toner transfer method
- Photolithography

3.1. Mesh and Ink Method
This method has become very popular over the years as it is one of the cheapest and easily reproducible methods of doing transfers onto copper clad boards. A print out from the design software is filmed and a mesh in made out of it. The mesh is a special kind of material that allows ink only through desired portions of it, specified during the filming process. A pattern similar to the designed circuit is made on the mesh with the desired circuit tracks being permeable. Sticker ink is used on the mesh to create the pattern. This is done by applying sticker ink on one side of the mesh and dragging it (the ink) across the mesh using a scrooge (a tool designed specifically for that purpose having a handle with a flat rubber side). Blank copper boards are available from electronic shops in various sizes, including double sided boards. But the surface must first be prepared before the mesh and ink transfer process. I prepare the copper board by using some fine grit sandpaper, a scotchbrite pad, or something similar. Sand the board in a circular motion to create very fine scratches and remove any copper oxide which has formed on the surface of the board [1].

After the board has been sanded, it must be cleaned to remove any traces of dirt or chemicals which may prevent the ink from sticking to the copper. It is also important that the cleaner does not leave any
chemical traces. Chemicals like acetone, nail polish remover, or methylated spirits evaporate any traces of themselves and are ideal for this purpose.

Fig. 6. A prepared Copper board

The mesh for the Automatic Car Security System is shown below.

Fig. 7. Automatic Car Security System’s Mesh

Sticker ink dries fast and leaves an etch-resistant coating on the copper board. This has the same effect of preventing the etchant from reacting with the board. Below is a transferred pattern from the mesh and ink method.

Fig. 8. Printed Circuit using Mesh and ink method
3.2. Direct Transfer

Here, you transfer your design to the board by using a permanent marker (most any "permanent" black marker will do) or rub-on transfer sheet to mark out where you want copper traces. The pen's ink resists the etchant, thus creating the desired circuit on the board. While cheap and simple, note that it is difficult to position the traces accurately (this can be a real problem if you are using ICs in your design). Also, since pens don't necessarily apply ink uniformly, there is a risk that some traces will be etched away since etchant can get to the copper through a thin layer of resist.

3.3. Press N Peel Paper /Toner Transfer Method

These two methods are similar in that they both make use of the fact that toner is plastic and will not react with etchants. The press n peel paper is quite expensive and can cost up to N400 each, however it gives a good result when used. An example of a transfer using the press n peel method is shown below.

![Press n Peel paper](image)

Blank copper boards are available from electronic shops in various sizes, including double sided boards. But the surface must first be prepared before the toner transfer process. After this the board is ready for the toner transfer. In order to transfer the toner from the paper to the PCB, you will need a hot cloths iron, a flat surface, some pliers, and a container of warm water. Toner has a reasonably high melting point, and providing the paper is carefully chosen it is best to set the iron temperature as high as it will go. This does not apply if using press n peel blue film which will melt and or shrink if heated to a high temperature. In this case follow the instructions on the sheet, otherwise, it is not uncommon for the paper to turn a light golden color during the toner transfer process.

When applying the iron, apply it with as much pressure as possible. While ironing, I typically rest my entire upper body on the iron. After the iron has been applied for a few seconds, the toner starts to melt and the paper sticks to the board. When this happens start moving the iron around the paper to ensure that all areas are properly heated and have pressure applied to them; otherwise areas under the steam-holes or other uneven surfaces may not transfer properly.

When the ironing is finished, a process which typically takes 3-6 minutes depending on board size, using pliers, drop the board in a container of warm water. Sometimes at this point the board will actually sizzle because of the heat. The board can then soak for 10-30 minutes depending on your patience and the paper used.

After the board has been soaked, it is time to carefully peel off the paper. Usually the paper will tear or won’t come off properly. It is only important to clear areas which will be etched, any paper on the
Toner traces can stay. To scratch off the paper either rub it off with your thumb or use a toothbrush. Toner is normally pretty sturdy and will not easily scrape off, so significant pressure can be applied. At this point, it is important to make absolutely certain that the traces are perfect. If you are unhappy with the transferred results, simply remove the toner with acetone and start again. Small scratches can be fixed with an etch-resistant marker. Areas which have transfer not desired should be scratched off with a very fine artwork knife or scalpel. If in doubt mark the traces with the etch-resistant marker, and make sure all component pads are free of paper, as next step is irreversible.

Fig. 10. Transferred patterns ready for etching

3.4. Photolithographic Method
In this case, a board is covered with a resist material that sets up when exposed to UV light. To make a board this way, you start by making a photo negative of the circuit which is clear where you want a circuit trace and opaque where you don't want a trace. After the photo negative is made from your artwork, it is placed onto the photo sensitized board, and is exposed to the UV. The UV light transmits through the clear portions of the negative and cures the photo resist. After that, the board is submerged into a developer bath that develops the traces on the board. The resist that is left is in the shape of the artwork that represents your circuit.

This approach is accurate and makes neat traces, and the photo negative can be used over and over to make additional boards. However, you need your own photo lab to do the board developing, and the entire process takes quite a bit of time.

4. ETCHING
Once the pattern has been transferred to the board, the unwanted parts will have to be removed to get the PCB done. The most popular way of doing this is by etching. Etching is a chemical process by which excess copper is removed by reacting it with chemicals which leaves the covered parts of the copper untouched. There are machines which remove the unwanted/excess copper directly from the PCB software. These machines are expensive and can only be purchased by companies having a very large capital base. Some common etchants are listed below.

- Ferric Chloride
- Ammonium Persulfate
- Sodium Persulfate
• Cupric Chloride.

4.1. Ferric Chloride (FeCl₃)

Copper is more readily attacked by ferric chloride (iron chloride), which is commonly used in concentrations of 28–45 percent [5].

The biggest benefits of Ferric Chloride are that you can buy it almost anywhere, it is cheap, and it is easy to use. You can buy it either pre-mixed in liquid form, or you can buy it in powder form that you later add water to.

The disadvantage of ferric chloride is that it is dirty. When brand new, the liquid is a brownish-yellow color that you could possibly barely see through if you shine a bright light directly through it. As soon as you start etching copper boards in it, it will turn a very dark brown, eventually almost black, that you cannot see through at all. The chemical will permanently stain anything it touches, so care should be taken while handling it.

When ferric chloride reacts with copper, it cools down. Cold ferric chloride will stop eating away at copper. Therefore, it is advantageous to continuously agitate the solution during the etching process. This will help move the cold ferric chloride away from the board, and replace it with warm ferric chloride that can continue etching away copper.

At room temperature, etching your board can take a really long time (about 1-2 hours), even with agitation. A better way to do it is to heat the ferric chloride a little bit during the etching process. If you get it too hot though, it will produce fumes and even start eating the resist layer of your PCB. It is recommended to heat it to a temperature no higher than 55°C (135°F).

The container to be used is plastic as any metallic container would be destroyed by the etchant. The usual practice is to put the plastic container in a bigger container where hot water can be added without diluting the etchant.

Ferric chloride keeps very well. As long as you keep it sealed up from contamination, you can re-use the same bath over and over again until the chemical is completely used up and will not etch any more copper.

4.2. Ammonium Persulfate

Ammonium Persulfate's biggest advantage is that it is transparent. It feels a little cleaner because it does not have the brownish-yellow color that Ferric Chloride has and it doesn't permanently stain as bad. It is not very easy to find, and you can buy it as a white powder or in crystal forms only. When you are ready to use it, you will mix it with water slowly, as it will produce a lot of oxygen bubbles. Freshly mixed, it will be a clear or slightly milky-clear liquid.

During the etching process, ammonium persulfate will produce tiny bubbles of gas where it reacts with copper. These bubbles will prevent the chemical from continuing to etch areas of the PCB. Therefore, it is necessary to agitate the solution during the etching process. It also seems to react very poorly at plain room temperature, so you will most likely need to heat the solution. Ammonium persulfate etches in just a few minutes with excellent results.

Once ammonium persulfate begins to eat away copper, it will turn a beautiful transparent blue color. The blue color will darken as the solution becomes more heavily loaded with etched copper. Ammonium persulfate seems to become saturated much faster than Ferric Chloride. The transparency of ammonium persulfate makes it much cleaner to use. Instead of having to pull the board out of the solution to check progress, you can simply shine a bright light through the solution and the board. As a result, you only need to pull the board once, after the etching process has finished [1, 5].

Ammonium persulfate is not as easy to store as ferric chloride, so it should be used up as soon as you've made it. If you do need to store it, keep in mind that the solution will continue to produce some oxygen gas. The gas will need to be able to vent safely instead of building up pressure and causing the container to burst. Below is a figure showing the etching process using Ammonium persulfate.
4.3. Sodium Persulfate
Sodium Persulfate is an alternative to ammonium persulfate that will work with etch resist pens.

4.4. Cupric Chloride
Cupric Chloride has the advantage that you can recycle the same bath over and over again indefinitely, just by adding a little HCl acid and bubbling air through the bath to regenerate the solution. It has a transparent green solution that would be just as clean as the Ammonium Persulfate. Together with the recycling potential, this could be the best etchant to use.

5. POST–PRODUCTION TASKS

There are a few things to be done after etching like cleaning the etched boards to ensure that the chemical reaction is stopped completely and other tasks like drilling and soldering.

5.1. Drilling
Etched boards are drilled to allow for component insertion. The components to be inserted determine the size of the bit to be used for drilling. Many types of drill are available for use but a battery-operated hand drill will be most suitable since it is lighter in weight and does not require power to be used. Care is usually taken during drilling to ensure that the holes are straight and not slanting and also to make sure that the holes are drilled on marked points on the board. And drilling is usually done on a wooden surface avoid breaking the bits. A typical example of a drilled board is shown below.
5.2. **Soldering**

This is a process common to most electronics engineers. However, most people are ignorant of some basic tips and tricks used to achieve a neat, stress-free soldering job. Before soldering, the resist material is removed using solvents like retarder, thinner or even methylated spirit. Some useful tips on soldering are outlined below [6].

- The iron should be prepared for use. This is done by filing the tip till it is about 45° to the length of the soldering iron. The iron is allowed to heat up and lead is applied generously over the filed surface. A film of lead is left behind afterwards to keep the tip from dirt that will inhibit smooth melting of the lead. See figure below showing a filed soldering iron and an unfilled one.

![Fig. 13 Soldering tips](image)

- Surfaces to be soldered must be cleaned to remove dirt, grease etc to allow for a neat coating of lead. The tips of the components to be soldered should be scraped using a razor blade to remove dirt. This ensures a good coat and also reduces the risk of making dry joints on the circuit boards.

- Care should be taken not to damage temperature sensitive components by leaving the soldering iron on them for longer than necessary. (this is usually the case when the surface and the components are not well prepared for soldering).
• Soldering should be done in a well illuminated room and away from fans and air-conditioners to avoid mistakes and allow the solder to melt and fuse properly.

• It is usual practice to solder IC bases first, followed by jumpers before components so that problems would not be encountered during component insertion.

5.3. Finishing
To improve the looks of the product, the drilled board can be sprayed to give it a professional touch. This also helps the board to last long as it will prevent the copper from rust and effects due to ageing. A sprayed board will look like one shown below.

Fig. 14. Finished Board

5.4. Construction
The construction of the project was realized by building and testing its different blocks separately i.e. the timing circuit and switching circuit. The timing circuit uses the 555 timer in a mono stable mode and the switching circuit uses 2N2222 as a switching transistor. The connection from one component to the other followed the specification on the circuit diagram. The whole components were skillfully lead-out on the PCB resulting in a compact and neat construction. A good fluxed 60/40 tin-lead solder alloy with a new 40W soldering iron having a nice tinned bit was used for the soldering [1, 7 8, 9 & 10].
Tools used include:
  • Soldering iron 40W
  • Square file
  • A soldering lead
  • Long nose pliers

6. CONCLUSION
The presented PCB design techniques are both efficient and economical. Students, researchers and small scale entrepreneurs in the field of electronic (or PCBs) will find this paper useful especially for its unique incorporation of economically proven local methods in PCB design, efficiently without compromising quality. Though EAGLE software was used to produce the PCB for this project, the knowledge can easily be adapted for any PCB design software, since the basics are similar irrespective of the software.
REFERENCE

[1] Orosun O. R., et al (2011) Development of Automatic Car Security System. Journal of Chemical, Mechanical and Engineering Practice. 3, 1-3.

[2] Orosun O. R. and Adamu S. S. (2012). Modeling and Controller Design of an Industrial Oil-Fired Boiler Plant, International Journal of Advances in Engineering & Technology 3(1), 534-541.

[3] CardSoft Inc. (2008) EAGLE V4.15® User’s Manual. “Getting started Guide for Windows”.

[4] Orosun O. R. and Adamu S. S (2013). Model Predictive Control of An Industrial Oil-Fired Boiler Plant, Zaria Journal of Electrical Engineering Technology 2(1), 39-56.

[5] Photoengraving. (2011). Encyclopædia Britannica. Ultimate Reference Suite. Chicago: Encyclopædia Britannica.

[6] Cooper, A. L. (1981) Electronics for Technical Level II. Staley Thorn Publisher Ltd.

[7] Orosun O. R. and Adamu S. S. (2014). Neural Network Based Model of An Industrial Oil-Fired Boiler System, Nigerian Journal of Technology Nsukka-NIJOTECH 33(2), 1-11.

[8] Orosun M. M., Orosun O. R. and Adamu S. S. (2016): Modeling and simulation of automatic generation control system for synchronous generator with model predictive controller. Zimbabwe Journal of Science & Technology pp 142 - 157 Vol.11 [2016] e-ISSN 2409-0360.

[9] Fredrick, F. D. & Robert, F. C. (1995) Operational Amplifiers and Linear Circuit (4th ed) Prentice Hall, Inc.

[10] Edward, H. (2002) Hughes Electrical and Electronics Technology (8th ed) Pearson Education.