Technical Support Bulletin

Faculty of Engineering

Issue No.14 May 2017

Printed Circuit Board, The Backbone Of Electronics

Authors: Naim Fuad, Electrical & Electronic Engineering Technical Support, FOE
Mohamad Irwan, Electrical & Electronic Engineering Technical Support, FOE
Noor Hashimi, Electrical & Electronic Engineering Technical Support, FOE

What does RoHS Compliant mean?

Have you ever noticed the RoHS Compliances logo on the back of a PCB? This mostly can be found in mass produced PCB and also in product description. Have you ever wonder what is the significance of this compliances impact towards us and also the company?

The RoHS stands for Restriction of Hazardous Substances. RoHS, also known as Directive 2002/95/EC, originated in the European Union and restricts the use of specific hazardous materials found in electrical and electronic products. All applicable products in the EU market after July 1, 2006 must pass RoHS compliance.

Any business that sells applicable electrical or electronic products, sub-assemblies or components directly to RoHS countries, or sells to resellers, distributors or integrators that in turn sell products to these countries, is impacted if they utilize any of the restricted materials.

The substances banned under RoHS are lead (Pb), mercury (Hg), cadmium (Cd), hexavalent chromium (CrVI), polybrominated biphenyls (PBB), polybrominated diphenyl ethers (PBDE), and four different phthalates (DEHP, BBP, BBP, DIBP).

WEEE or Waste from Electrical and Electronic Equipment compliance aims to encourage the design of electronic products with environmentally-safe recycling and recovery in mind. RoHS compliance dovetails into WEEE by reducing the amount of hazardous chemicals used in electronics manufacture.

To put another way, RoHS regulates the hazardous substances used in electrical and electronic equipment, while WEEE regulates the disposal of this same equipment.

1. Introduction

The birth of electronic devices changes the way we see the world today. Back in the days, machinery are powered by gears and manual operation. Now, electronic devices are capable of doing complex task to help ease our live. The backbone for every electronic devices nowadays are the Printed Circuit Board or PCB where all the electronic component is placed and held together.

The PCB fabricating services are available here in The University of Nottingham under the department of Electrical and Electronics Engineering. This service are not intended solely for Electrical and Electronics Engineering Students in their mini project, it is also available for Final Year Project, undergraduate and postgraduate students that use electronics in their research.

2. Designing a PCB

The PCB designs can be created both manually and automatically. Manual layouts are created with the help of CAD drafting, and the automatic router helps in the creation of the designs automatically. The designers usually prefer the manual way of designs, since they can implement their own ideas and techniques in them.

With the increase in demand of electronic devices, the number of PCB services is on the rise. Such designers generally offer a complete package of the PCB design services. This includes the PCB editor, the design capture technology, an interactive router, a constraint manager, interfaces for manufacturing CAD, and the component tools where the PCB editor edits the layers in the PCB, both single and multilayered.

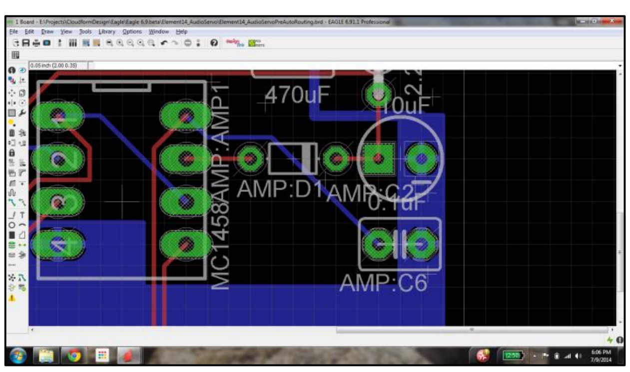

Fig. 1 Designing PCB by using Eagle

3. Composition of PCB

A PCB normally consist of layers of silkscreen , solder mask, copper and substrate. These layers are alternated and laminated together through the process of heating and adhesive.

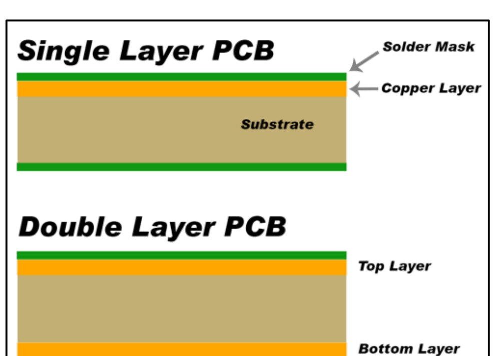

Fig. 2 Comparison Between Single and Double Layered PCB

3.1 Substrate

The base material, or substrate is usually fiberglass which commonly called as FR4 in order to give its rigidity and thickness. FR4 consist of a woven glass fibre mesh soaked in organic polymer (epoxy) with copper layer laminated. The “FR” in the name stands for flame retardant and denotes the safety of flammability of the FR4 is in compliance with the standard of UL94V-0. There are also flexible PCB that are built on flexible and high temperature plastic such as Kapton.

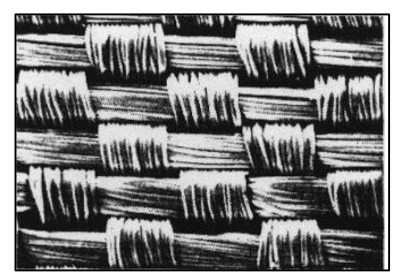

Fig. 3 Woven fibre glass substrate under electron microscope

3.2 Copper

The next layer is a thin copper which is laminated to the board with heat and adhesive. The layer of copper can either be only single layered to 16 layers and more. The copper thickness can vary and specified by weight. The vast majority of PCB have 35 micrometer of copper per square foot but some PCB that handles very high power may use up to 70 micrometers or more.

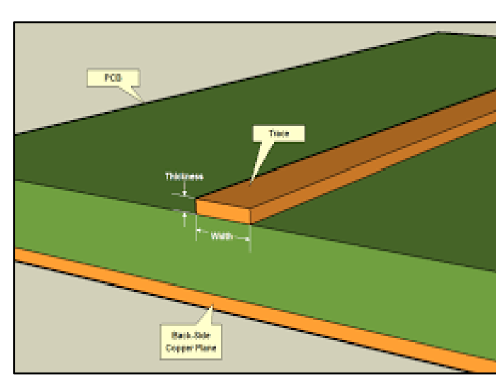

Fig. 4 Copper track

3.3 Soldermask

The layer on top of copper is called soldermask which made up from a thin lacquer-like layer of polymer. It is overlaid on top of copper layer to insulate the copper traces from accidental contact with other metal, solder jumpers or conductive bits.

Apart from that, this layer also helps during soldering process to protect against oxidation of the copper traces on the board. The soldermask is traditionally green colored but nowadays it comes with a variety of colors.

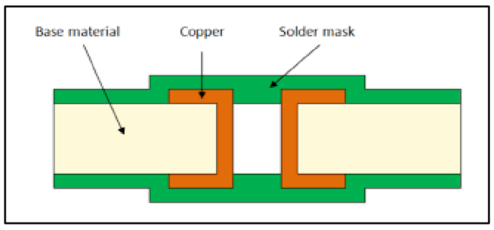

Fig. 5 Soldermask

3.4 Silkscreen

The white silkscreen layer is applied on top of the soldermask layer. The silkscreen adds letters, numbers, and symbol to the PCB that allow for easier assembly and indicators for students to better understand the layout of the circuit. The silkscreen is commonly in white color but any color can be used.

Fig. 6 Silkscreen

4. Type of PCB

PCBs can be classified into different categories according to different classification standards. Based on the number of layers, PCBs can be classified into single-layer PCBs (also called single-sided PCBs), double-layer PCBs (also called double-sided PCBs) and multi-layer PCBs.

Based on the substrate material, PCBs can be classified into rigid PCBs, flexible PCBs and flex-rigid PCBs. Each type of PCBs is applied in different fields owing to their advantages together with consideration of their disadvantages. This topic will cover the commonly used type of PCB for student in University of Nottingham.

4.1 Single Sided PCB

This type of printed circuit board contains just one layer of substrate, or base material. One side of the substrate is covered with a thin layer of metal, typically copper because it's an excellent electrical conductor. Usually, a protective solder mask sits on top of the copper layer, and a final silkscreen coat may be applied to the top to mark parts of the board.

A single-sided PCB has electronic components and circuits on only one side. This type of board works best for simple electronics, and at-home hobbyists often design and construct this type of board first.

Single-sided boards tend to cost less to mass-produce than other board types. But despite this low cost, they're used infrequently because of their inherent design limitations

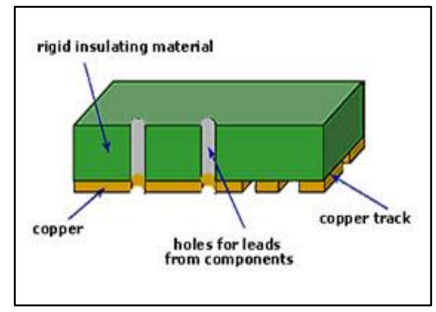

Fig. 7 Single Sided PCB with copper track and holes

4.2 Double Sided PCB

Double-sided PCBs are much more common than single-sided boards. Both sides of the substrate have metal conductive layers, and parts are attached to both sides as well. Holes in the board allow circuits on one side to connect to circuits on the other side.

Double-sided circuit boards connect the circuits on each side using one of two methods: through-hole technology and surface mount technology. Through-hole technology involves feeding tiny wires, known as leads, through the holes and soldering each end to the appropriate component or circuit.

Unlike through-hole technology, surface mount technology does not use wires. Instead, many small leads get soldered directly onto the PCB. Surface mount technology allows more circuits to be made in a smaller space on a board, meaning the board can perform more functions, usually at a lower weight and at faster speeds than through-hole boards allow.

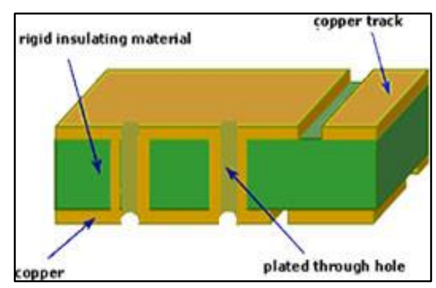

Fig. 8 Double Sided PCB with copper track and holes

4.2 Multi-layered PCB

Multilayer PCBs expand on the technology used in double-sided boards. They have several layers of substrate boards, and insulating materials separate the individual layers. As with double-sided boards, components on multilayer boards can connect to each other through holes, or vias , in the board.

The multilayer design saves even more space than a double-sided design. Typical multilayer boards have 4, 6, 8, or 10 layers, but they can have more, depending on the demands of the product the board is intended for. Multilayer PCBs are found in equipment like computers, servers, medical machinery, and hand-held devices.

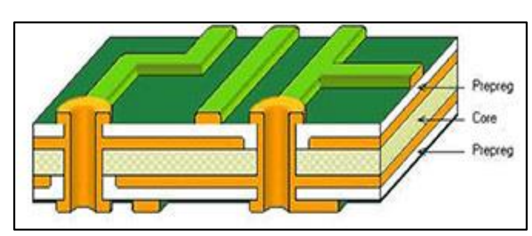

Fig. 9 Multi-layered PCB with copper track and holes

5. Process Workflow for PCB Fabrication

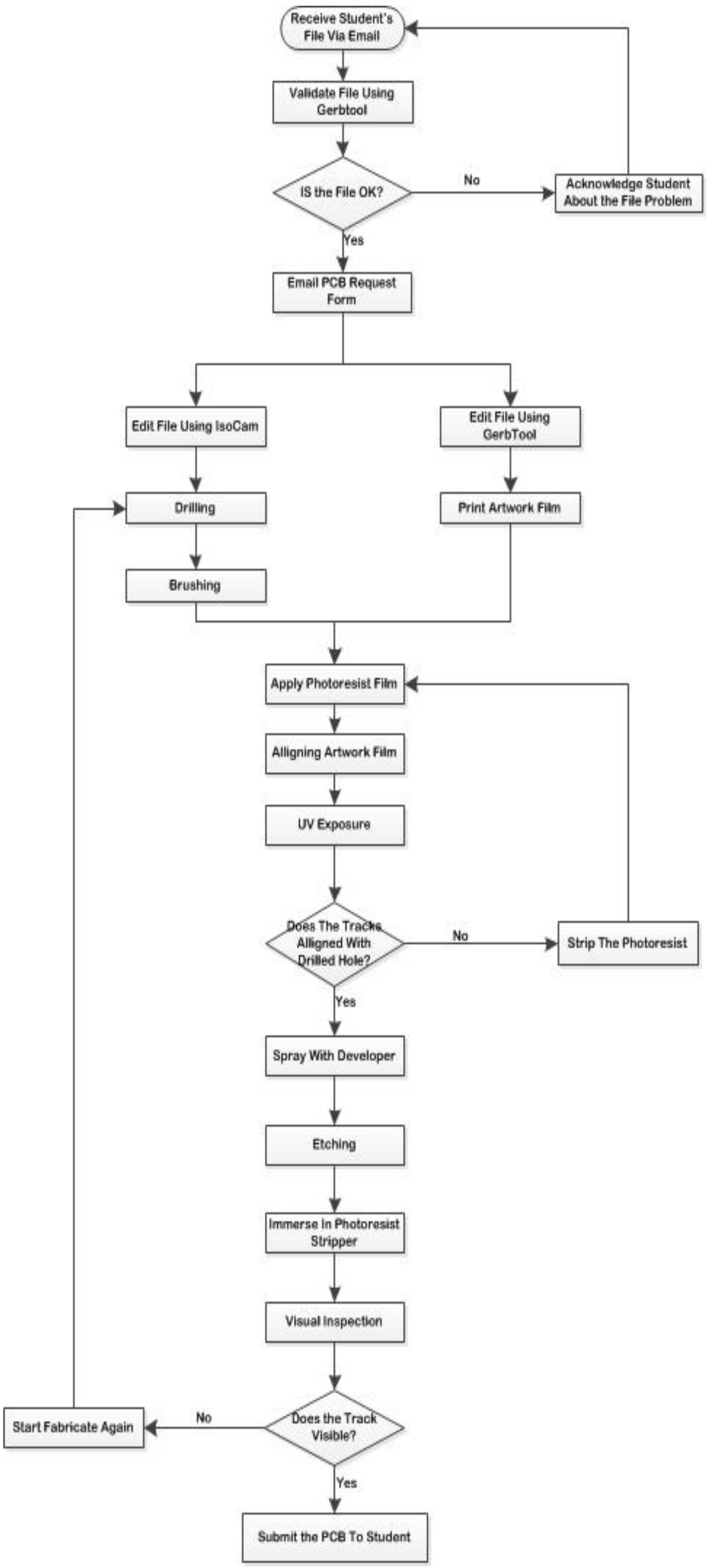

Chart 1. Process Workflow of PCB Fabrication in The University of Nottingham

6. Fabrication Process

6.1 Printing Artwork Film

The Gerber data for the part is used to plot film that depicts the traces and pads of the board's design. The photo tools or artwork include solder mask and legend or nomenclature as well as the copper features. This film is used to place an image on the resist.

A specially formulated translucent film for the production of high resolution PCB artworks directly from any Laser printer. The film will also accept copier toner enabling usable artworks to be produced from pre-printed originals. The tracks printed on the film is to block UV light so that the opaque color on the film will etched away during the etching process.

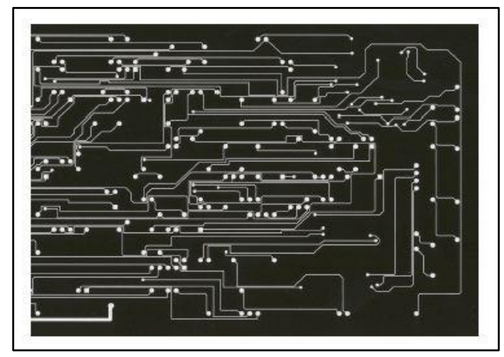

Fig. 10 Artwork film

6.2 Drilling

The drilling command is executed through the Bungard RouterPro application to send data information from the PC to the drill machine.

All the drilling files are edited through IsoCam to combined all the PCB designed into a single board layout. The data is then translated to Bungard RoutePro application to execute the drilling command wirelessly to the drilling machine.

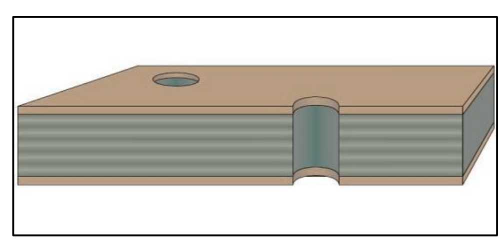

Fig. 11 Copper Board with Drilled Hole

6.3 Brushing

The drilled board is later go through a brushing process where the board will be brushed to remove any impurities such as oils (fingerprint) or dust. This process is important so that any impurities will not hinder with the end result of the copper traces after etching process

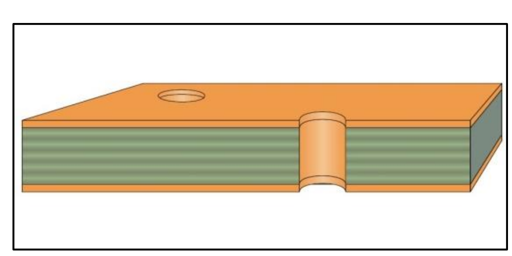

Fig. 12 Copper Board After Brushing

6.3 Photoresist Laminating

Next, the clean panel receives a layer of photo-sensitive film called the photo resist. The photo resist comprises a layer of photo reactive chemicals that harden after exposure to ultra violet light.

This ensures an exact match from the photo films to the photo resist. The films fit onto pins that hold them in place over the laminate panel.

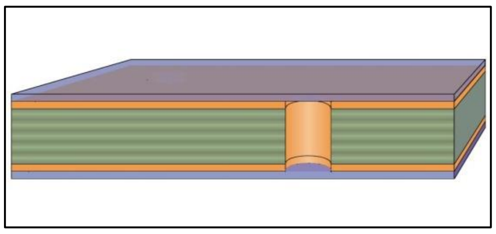

Fig. 13 Copper Board With Laminated Photoresist

6.3 Aligning Artwork Film

The artwork film will act as a barrier so that no UV light will be exposed to the track artwork. The drilled holes can be made as a pilot hole to aligned the artwork with the board.

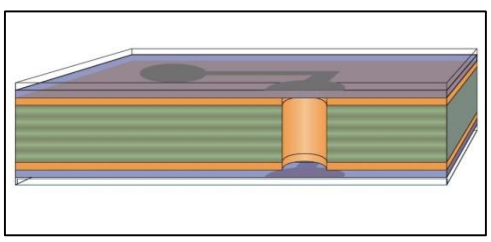

Fig. 14 Copper Board With Artwork Film Aligned

6.4 UV Exposure

The film and board line up and receive a blast of UV light. The light passes through the clear parts of the film, hardening the photo resist on the copper underneath.

The black ink from the plotter prevents the light from reaching the areas not meant to harden, and they are slated for removal. The aspect of light control is vital during this stage, and only yellow light floods the interior of the room. Yellow light carries the least UV level wavelengths.

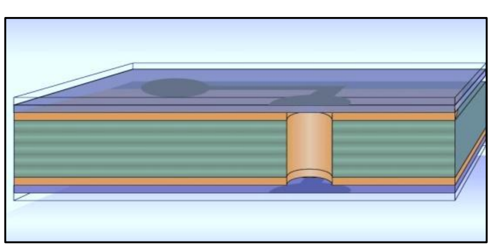

Fig. 15 Copper Board Under UV Exposure

6.4 Developing Unexposed Copper Track

After the board becomes prepared, it is washed with an alkaline solution that removes any photo resist left unhardened. A final pressure wash removes anything else left on the surface. The board is then dried.

The product emerges with resist properly covering the copper areas meant to remain in the final form. A technician examines the boards to ensure that no errors occurred during this stage. All the resist present at this point denotes the copper that will emerge in the final PCB.

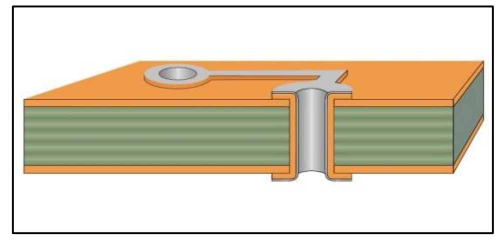

Fig. 16 Developed Copper Board with Photoresist Traces on Top of Copper Track

6.3 Etching Process

With the photo resist removed and the hardened resist covering the copper we wish to keep, the board proceeds to the next stage: unwanted copper removal. Just as the alkaline solution removed the resist, a more powerful chemical preparation eats away the excess copper. The copper solvent solution bath removes all of the exposed copper. Meanwhile, the desired copper remains fully protected beneath the hardened layer of photo resist.

Not all copper boards are created equal. Some heavier boards require larger amounts of copper solvent and varying lengths of exposure. As a side note, heavier copper boards require additional attention for track spacing. Most standard PCBs rely on similar specification.

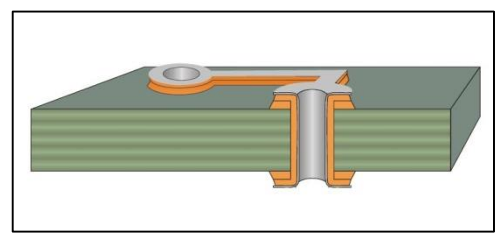

Fig. 17 Copper Board With Copper Track and photoresist layer

6.4 Strip Photoresist

Lastly, to remove the photoresist film, the board is then immersed in photoresist stripper so that the copper track remain on the board.

Visual inspection is made meticulously at this point to make sure that all the copper tracks are visible and connected.

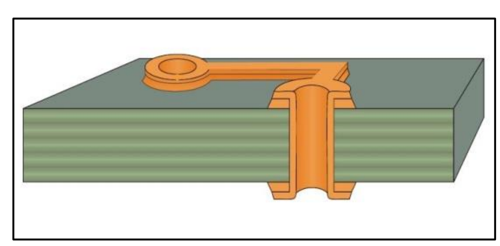

Fig. 18 Final Product for PCB Fabrication

7.0 References

1)Verdant Electronic, The Occam Process An Energy efficient Approach to Energy-Electronic Assembly, Interconnection, and Data Transmission Management, Joseph Fjelstad
2)University of Oslo FYS4260/FYS9260: Microsystems and Electronics Packaging and Interconnect – Printed Circuit Board
3) Advanced Circuit. How To Build A Printed Circuit Board
4)http://en.wikipedia.org/wiki/Printed_circuit_board