



# SYSTEM-IN-PACKAGE



## Prototyping and Test Board Assembly

The prototyping stage of a printed circuit board is the moment in which the idealized and designed circuit materializes, therefore it is a stage that requires care and attention so that everything that was designed is achieved.

### PCB Preparation

There are a few ways to make a printed circuit board, from simple to more sophisticated or industrial processes. This tutorial will cover some methods for transferring a design made in an electronic CAD editor to a flat copper board.

- **Thermal Transfer Method**

In this method the positive of the circuit is created, which is printed on a special paper. This paper is placed on the copper plate and through a thermal process (heating), the tracks printed on the paper are transferred to the surface of the copper. This tutorial will focus on this process, as it is the most widely used in handcrafting because it is simple, easy and gives a good result.

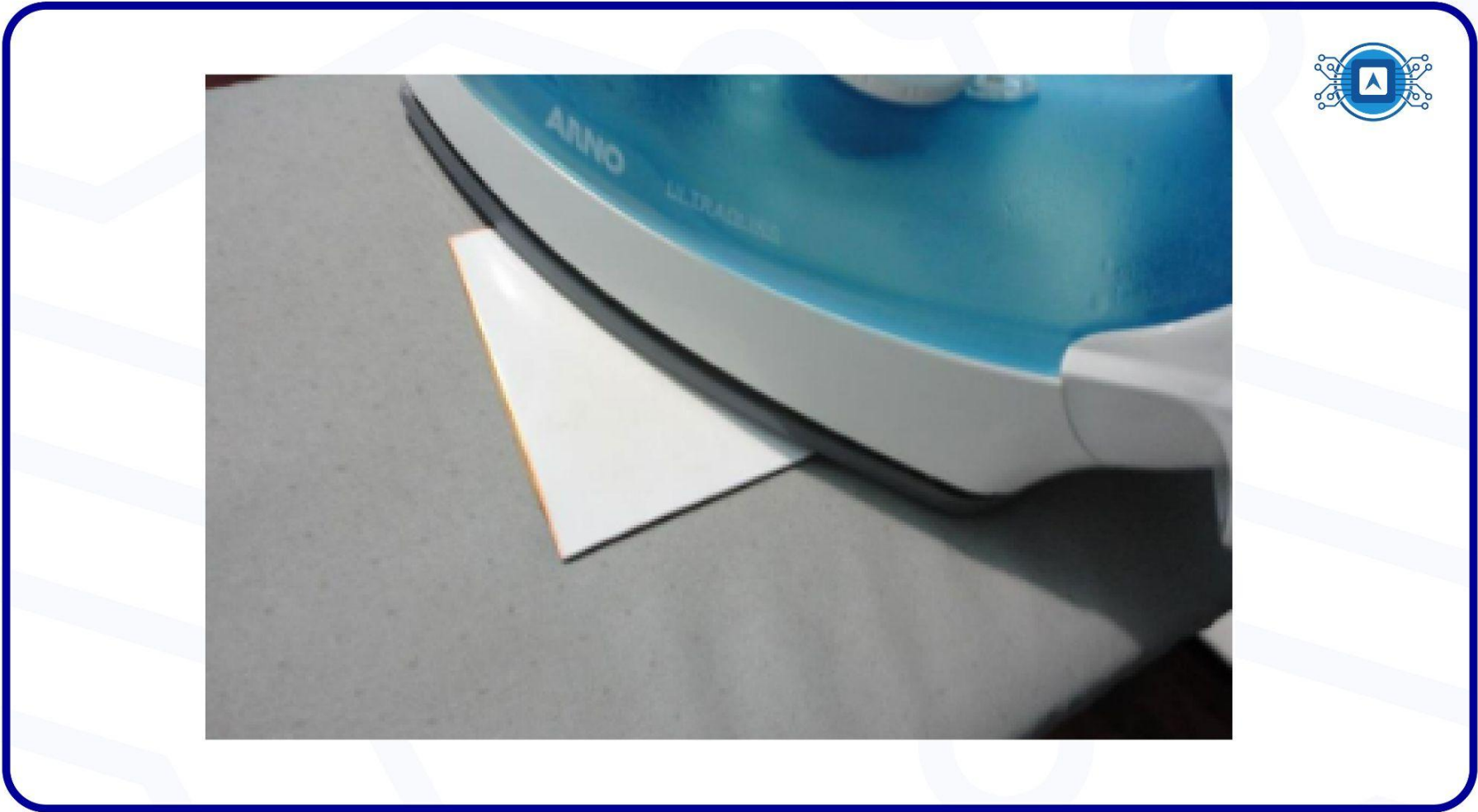


Image 01: Thermal Process of making PCB. Source: *The Author*.

## - Photographic method

In this method, a negative of the circuit is created and printed on special paper. This paper is placed on the board and a light-sensitive product (photolitho) is applied. This product in contact with light will react and mark the tracks, which will then be protected from corrosion. This process resembles the photo developing process used in the old days.

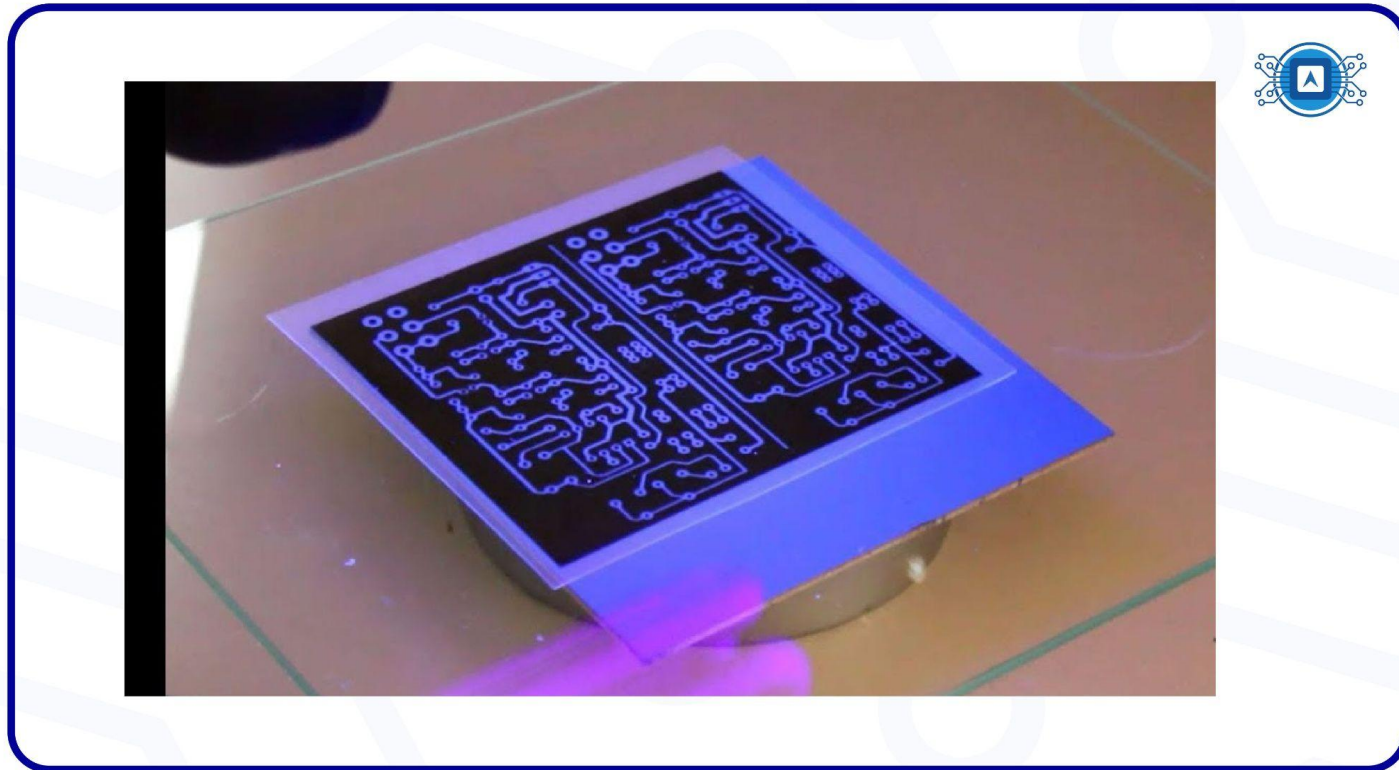


Image 02: Photographic process of making PCB.

Source: Screenshot from the video Tutorial: Circuito impresso fácil e perfeito pelo método fotográfico <<https://www.youtube.com/watch?v=IA9PxklUJZQ>>.

## - Dry Film Method

The most common printed circuit board production process is to place the dry film over the film with the circuit pattern and expose/reveal it in the UV machine. The dry film uncovered by the circuit pattern is hardened, while the covered dry film can be removed. The dry film can be further classified into photopolymerization type and photodecomposition type. The photopolymerization type dry film solidifies under light irradiation of a specific spectrum and is changed from a water-soluble substance to an insoluble type.



Image 03 - Dry Film used to make PCB. Source: *The Author*.

## - Method using a milling machine

A milling cutter is a mechanical tool prepared for machining parts or building others. To process a PCB, it is necessary to remove the copper surface of the board using drills and end mill bits with slots specifically for this purpose. In building a copper track, one or two lateral lines of the material are removed, leaving the central part in a rectangular shape. The part of copper that remains on the PCB constitutes copper trails, joining up to the pad islands.

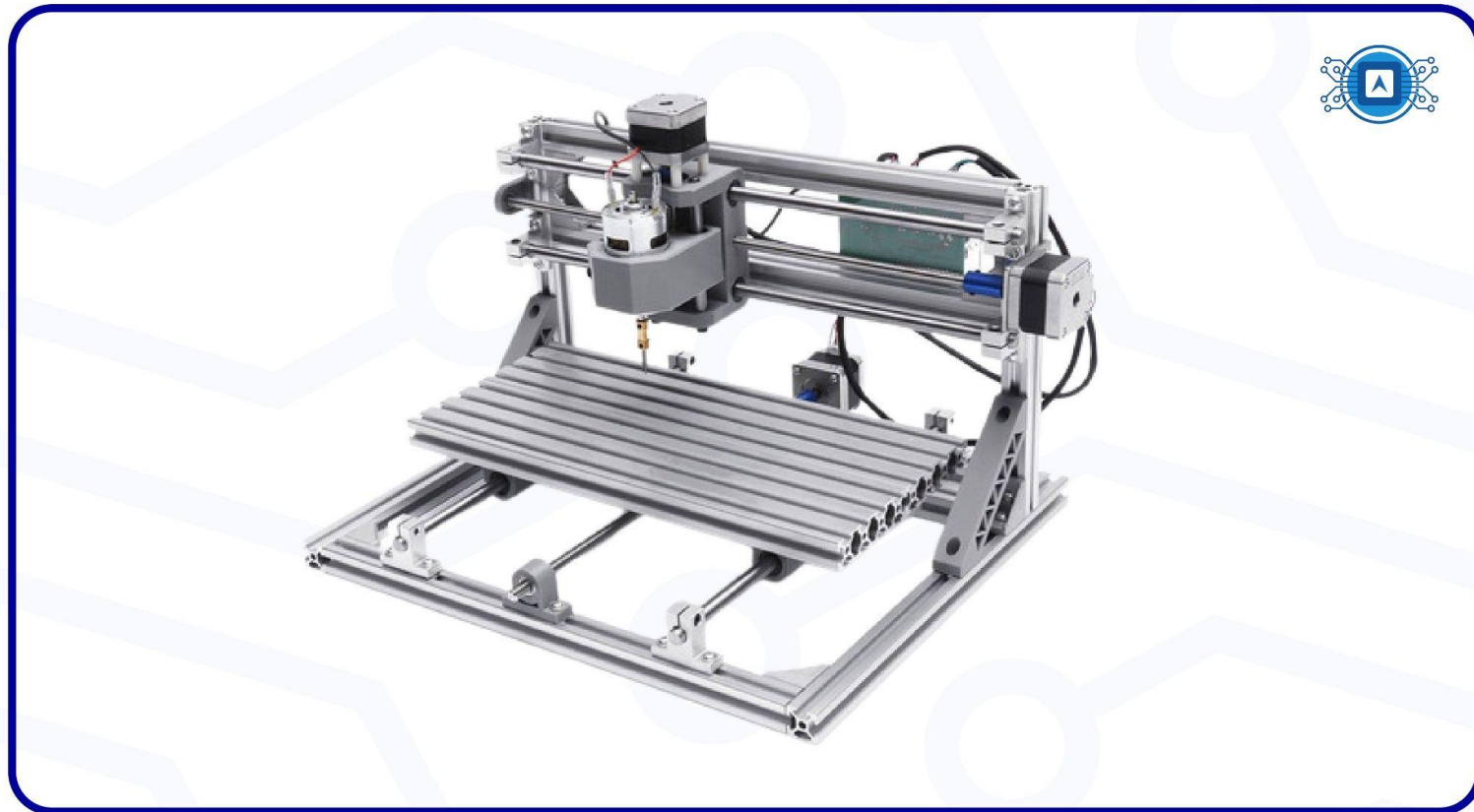


Image 04: PCB Milling machine. Source: The Author.

## Types of PCBs

### - Phenolite

Phenolite sheets are made from a mixture of a phenolic resin and shredded paper or sawdust. It is light or dark brown in color, which varies according to the proportion of raw materials. This mixture is molded and pressed into sheets of varying thickness. One of the problems in using phenolite is that its composition has a cellulose base, which absorbs some moisture from the environment, which can harm the insulating characteristics and can cause the boards to bend.

To try to get around this problem, in the 1960s fiberglass sheets were developed. These plates are made from epoxy resin and a thin fiberglass blanket, and their use makes fiberglass plates inert to water. However, this type of board becomes more difficult to cut and drill. Fiberglass boards are about 30% more expensive than phenolite, but most electronic equipment is made with fiberglass boards, and phenolite boards are restricted to lower quality designs or when using handcrafted manufacturing techniques.

### - Fiberglass (FR4)

Fiberglass or FR4 is a material that presents superior characteristics to phenolite, however, its application is not always possible in certain projects, due to its high cost if compared to phenolite. Table 1 shows a comparison between phenolite and fiberglass. Although the characteristics of phenolite are inferior to fiberglass, phenolite is still widely used due to its lower cost and the fact that its characteristics meet the needs of most projects.

| Type       | Cost | Mechanical Resistance | Thermal Resistance | Isolation |
|------------|------|-----------------------|--------------------|-----------|
| Phenolite  | Low  | Average               | Low                | Low       |
| Fiberglass | High | High                  | High               | High      |

Table 01 - Comparison between Phenolite and Fiberglass. Source: *The Author*.



## Using Gerber Files

Gerber files are usually produced by a PCB CAD tool and are sent to a PCB manufacturer who uses the files to manufacture printed circuit boards (PCBs).

There are two standards that define Gerber files: an older one called RS-274D, and a newer one called RS-274X. Gerber files generated in the old format (RS-274D) do not have opening information, which must be provided separately. The newer standard (RS-274X) incorporates opening information into the file.

When creating Gerber files using your PCB tool, be sure to export RS-274X files if you want to view the files with gerbv. In the Gerber format, the different layers of a PCB (for example silk top, metal top, inner layers, etc.) are kept in separate files. PCB stacking information (that is, layer order, thickness, etc.) is not captured by the Gerber files; it is up to the user to provide this information to gerbv and to their PCB manufacturer.

### Gerbv

Gerbv is a viewer for Gerber RS-274X files, drill files, and pick-and-place files.

Gerbv is a native Linux application and runs on many common UNIX platforms. A Windows version is also available. Gerbv is free / open source software.

Link to download Gerbv: <https://sourceforge.net/projects/gerbv/files/>

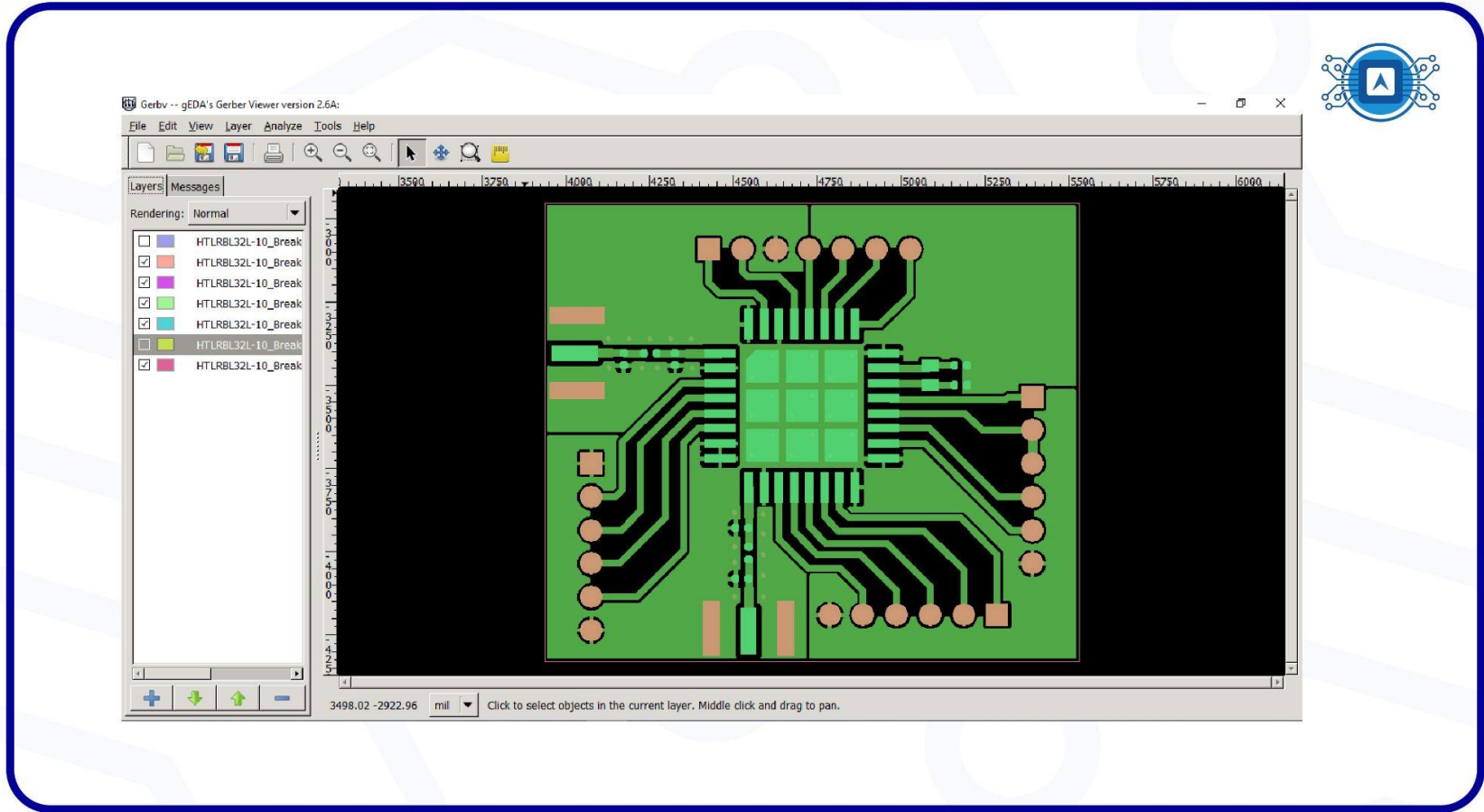


Image 05: Gerbv. Source: Screenshot by the author.



## Confection

For the cases of confections that use thermal, or photographic image transfer processes, the process of plate corrosion is used, and we will give a brief summary of this process here:

In a glass, plastic or ceramic container, place the plate attached by a wire or string and pour in the Iron perchloride until it completely covers the plate. Use the string to move the plate in the solution. It is important to remember that we should avoid direct contact with perchlorate, which, although it is not toxic, easily stains clothing and corrodes metals if exposed for long enough.

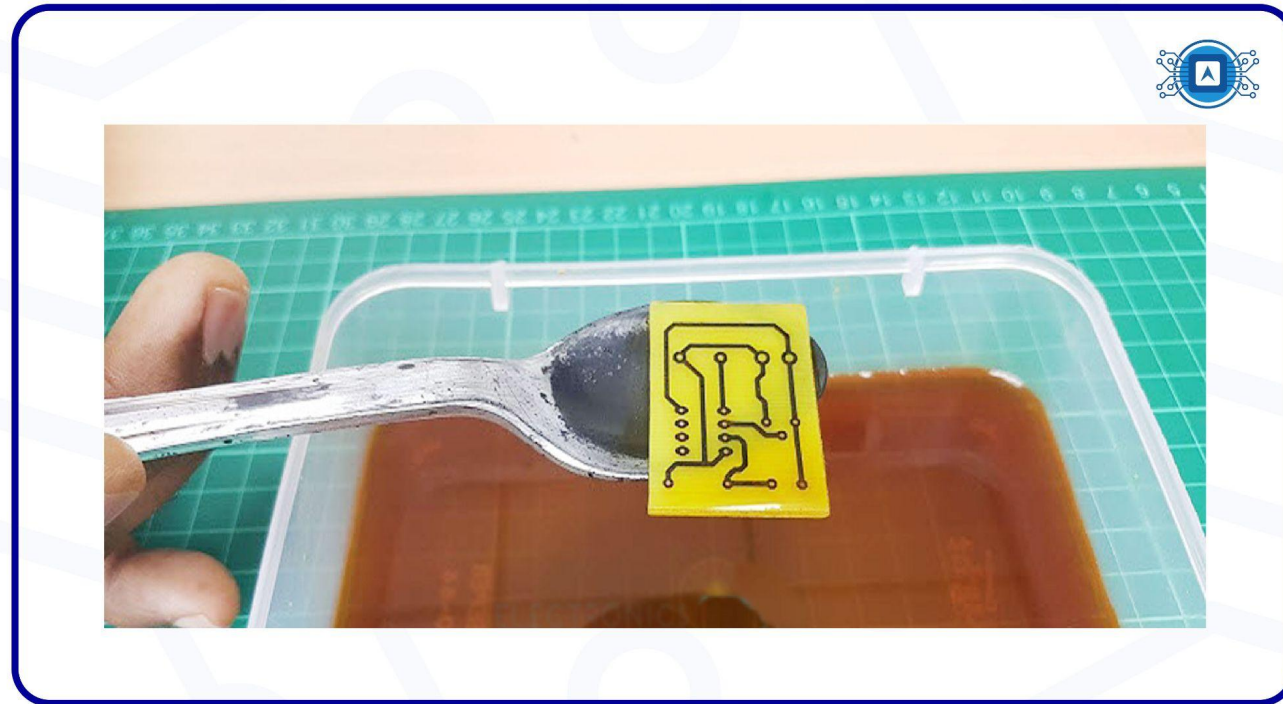


Image 06: Corrosion process on PCB. Source: *The Author*.

## Component Welding:

### - PTH (Pin Through Hole) Components

Now that we have the board ready, we will demonstrate some soldering techniques to make a good solder. First, we must clean the tip of the soldering iron, usually using wet vegetable sponges; do not use abrasive products as these can damage the tip of the soldering iron.

After cleaning, tin the tip, this will facilitate the soldering process.

Next, position the component on the insulating side of the board, as shown in image 6:

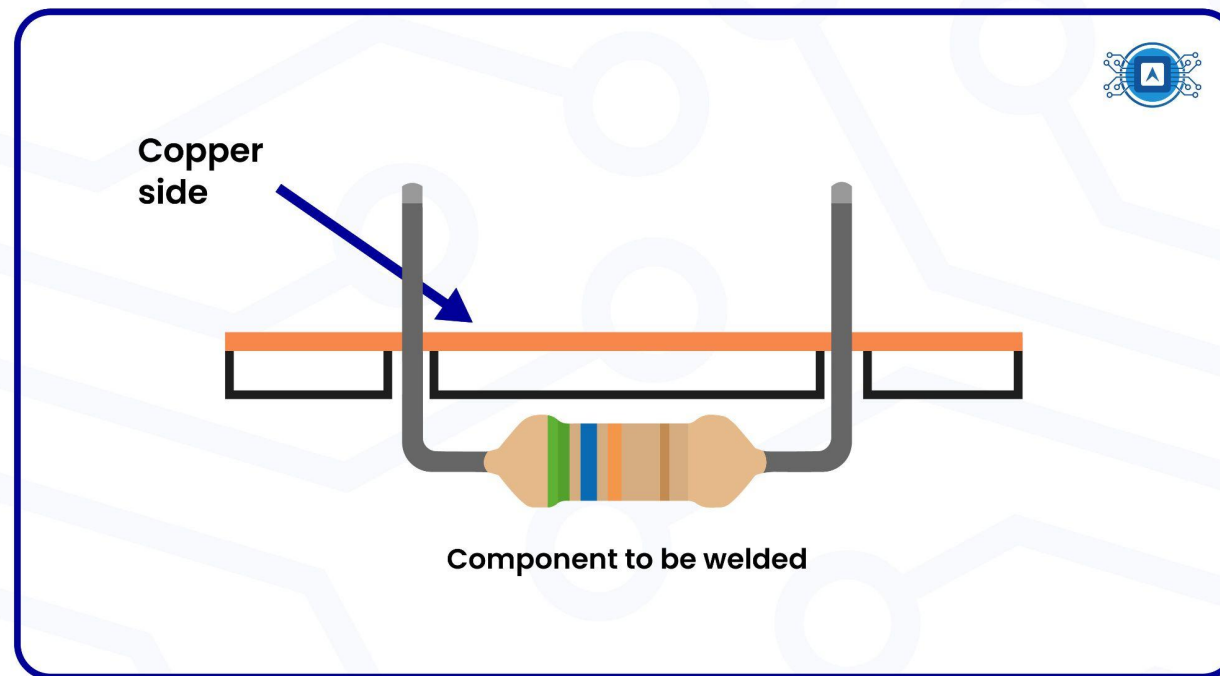


Image 07: Correct PTH welding position. Source: *The Author*.

The next step is to position the tip of the soldering iron at about a 45° angle and keep the component's terminal in contact with the hot iron for a few seconds.

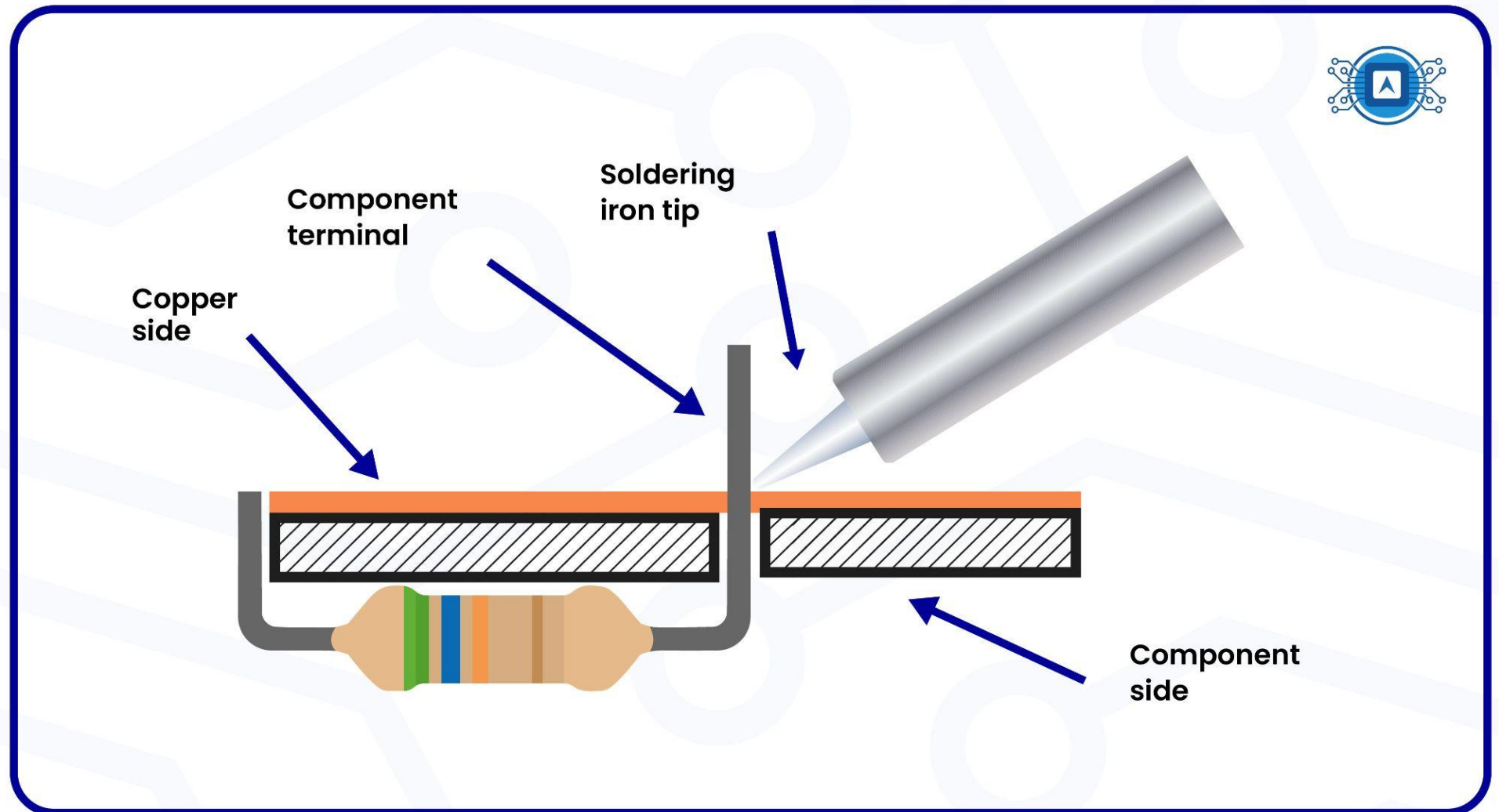


Image 08: Correct PTH (1/2) welding position. Source: *The Author*.

Then touch the solder (the tin) to the component terminal, from the opposite side of the soldering iron tip. The solder will melt.

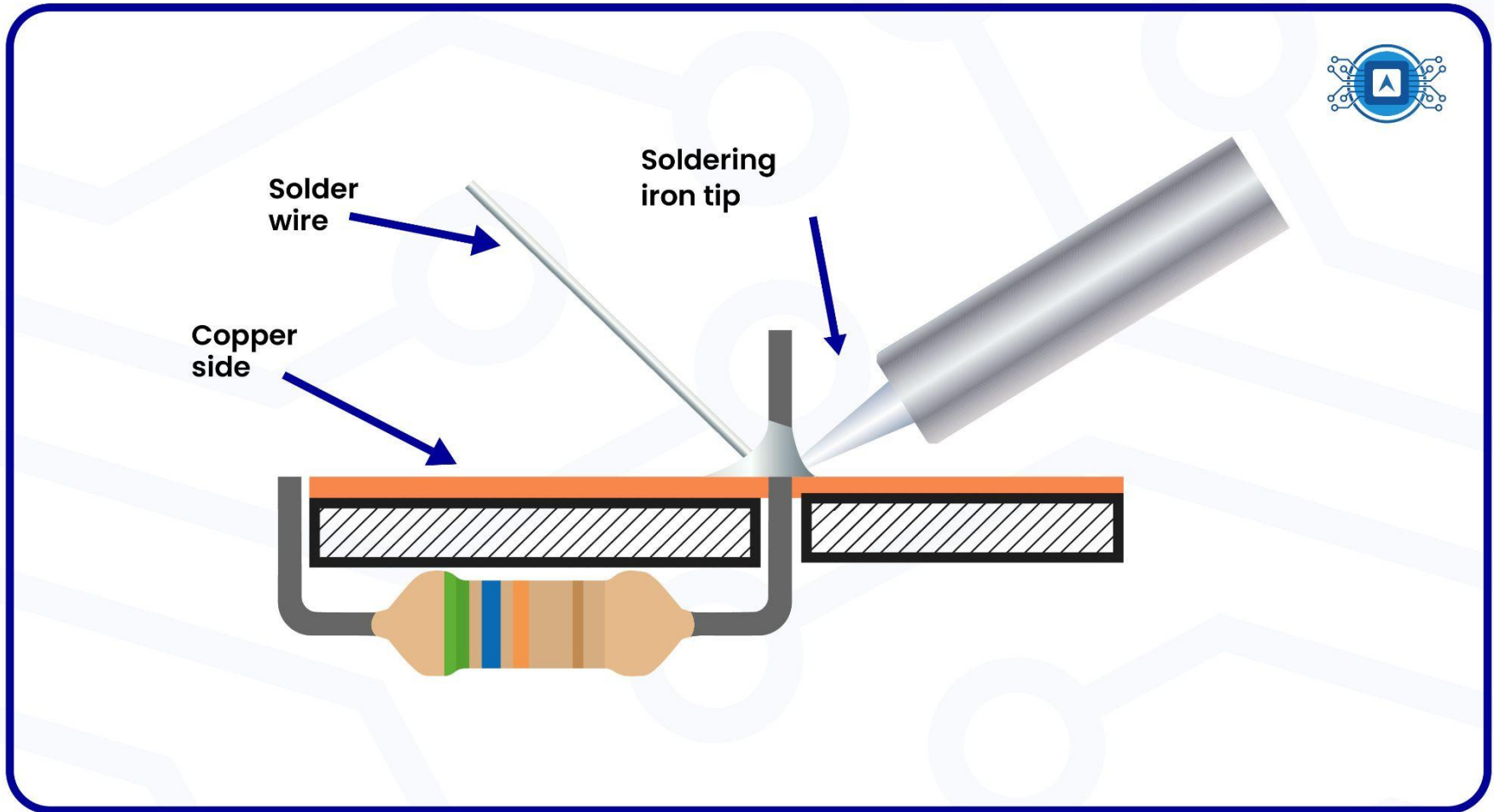


Image 09: Correct PTH (2/2) soldering position. Source: *The Author*.

Cut the excess from the terminal.

## SMD (Surface Mounted Device) Soldering

Despite the enormous advantages that this technology offers on an assembly line over traditional pin-through-hole assembly, there is the need to completely change the technology and therefore the equipment that will be used.

Regular components are supported by their terminals in an insertion through-hole process while SMDs are supported by the solder joints themselves, as shown in the image below.

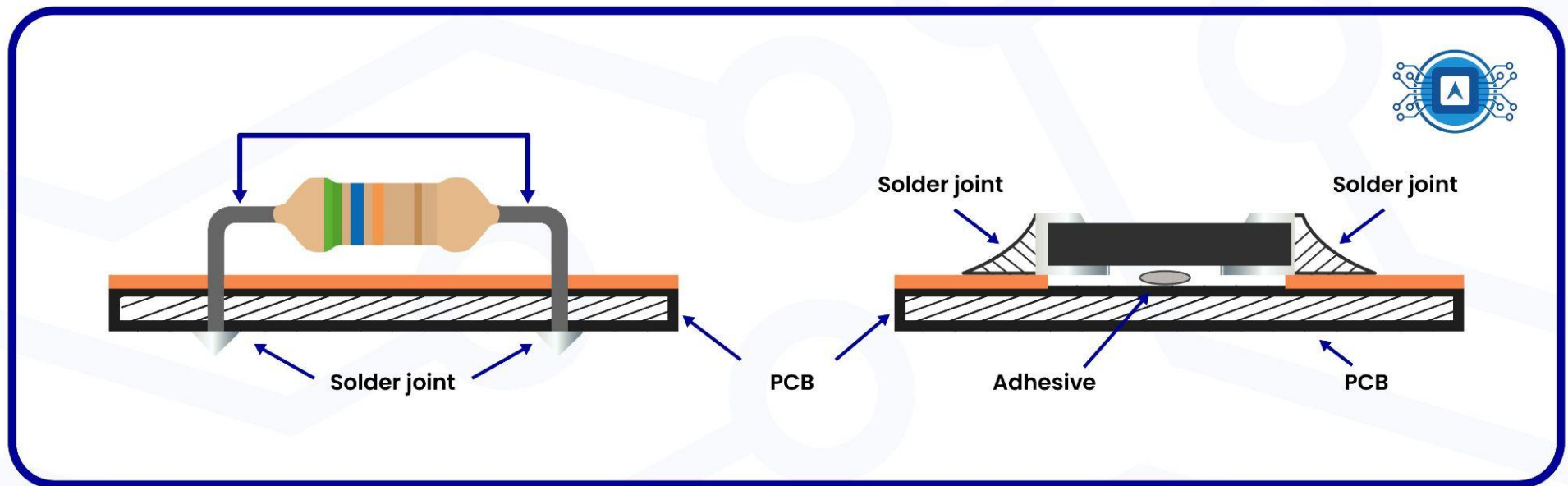


Image 10: Comparison of PTH(left) and SMD (right) welding. Source: *The Author*.



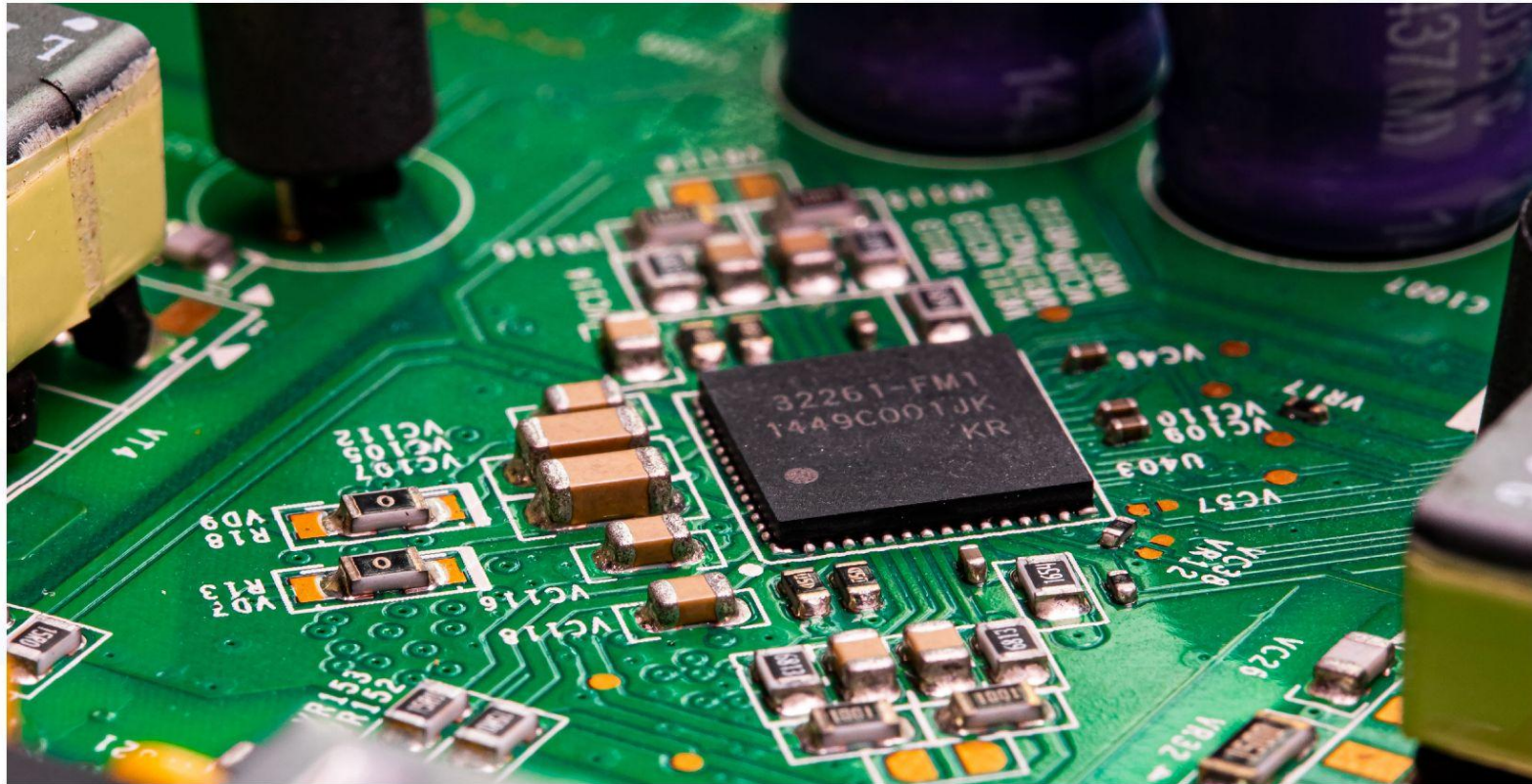


Image 11: SMD components soldered onto PCB. Source: Shutterstock.



## References

CIRCUITS, Instructables. **How to Make PCB at Home**. Available at: < <https://www.instructables.com/How-to-make-PCB-at-Home/> >. Accessed on July 25th 2022.

SCIENCEBUDDI. **HOW to Solder Electronic Components: Electronics Primer**. Available at: < <https://www.sciencebuddies.org/science-fair-projects/references/how-to-solder> >. Accessed on July 25th 2022.

SOLUTIONS, Orcad PCB. **PCB Fabrication Techniques**. In: Blog Cadence , PCB DESIGN & ANALYSIS. [S. l.], 2022. Available at: < <https://resources.pcb.cadence.com/blog/pcb-fabrication-techniques-2> >. Accessed on July 25th 2022.

## Additional readings

Several methods that can be used to manufacture boards, some simpler others more complex. You can find them using the links below:

Thermal Process for PCB building. < <https://www.youtube.com/watch?v=1bJoTFhJ5CY> >. Accessed on July 25th 2022.

Using Milling machine for PCB building. < <https://www.youtube.com/watch?v=zRdJrE80Vjk> > Accessed on July 25th 2022.