

Constructing the TNT-InDiscreto on a Printed Circuit Board

In a Printed Circuit Board (PCB) the connections between electronic components are made by copper tracks that adhere to an insulating laminated substrate.

To produce such tracks one uses a subtractive technique: one removes the excess copper through the action of a chemical substance capable of corroding it, having cleverly protected, by means of a paint resistant to the attack of corrosive substances, the course of the tracks that will make the connections of the circuit.

Direct transfer is a very effective method of producing simple PCB circuits: the protective paint is applied manually, using a suitable marking pen.

NECESSARY MATERIALS

Everything required can be found in a well-appointed electronics store. The following are needed:

- A single-sided plain copper PCB for the power supply: standard dimensions 10x16 cm
- A double-sided plain copper PCB (i.e. with copper on both sides) for the amplifier. Both channels of the amplifier can be neatly accommodated on a standard sized PCB of 10x16 cm. To separate the two channels further, a larger PCB would be necessary.
- A bottle of ferric chloride (or some oxidising acid): we're talking about a substance capable of corroding copper (among other things), which is used for the etching, i.e. to remove the excess copper from the substrate leaving only the tracks that form the circuit.
- A sheet of blue carbon paper to trace the pattern of the tracks on the copper. It's important that the carbon paper be blue and greasy: other types of carbon paper don't work on copper.
- A marking pen with which to draw the tracks with paint that protects the underlying copper from corrosion. There are various types and sizes to be found in stores; a DALO33 would certainly be fine. Any other marking pen or paint that resists acid attack would also be suitable.

PROCEDURE

1. *Polish the copper layer* of the PCB with a kitchen scourer or with superfine sandpaper, the aim being to remove any oxide which might stop the protective paint from adhering properly.
2. *Trace the pattern* of the circuit tracks on the copper using the carbon paper and *apply the protective paint* copying the design with the DALO33 marker pen - or whatever other type of marker pen you have chosen to use. Wait till the paint is properly dry before proceeding (20 minutes should be enough)

N.B.: to protect the side of the amplifier circuit board that will function as a ground plane it is possible to use adhesive packing tape (plastic not paper), covering the entire surface with several overlapping layers so that you leave no spaces between the joins. Pay particular attention that the tape adheres firmly at the edges of the board.

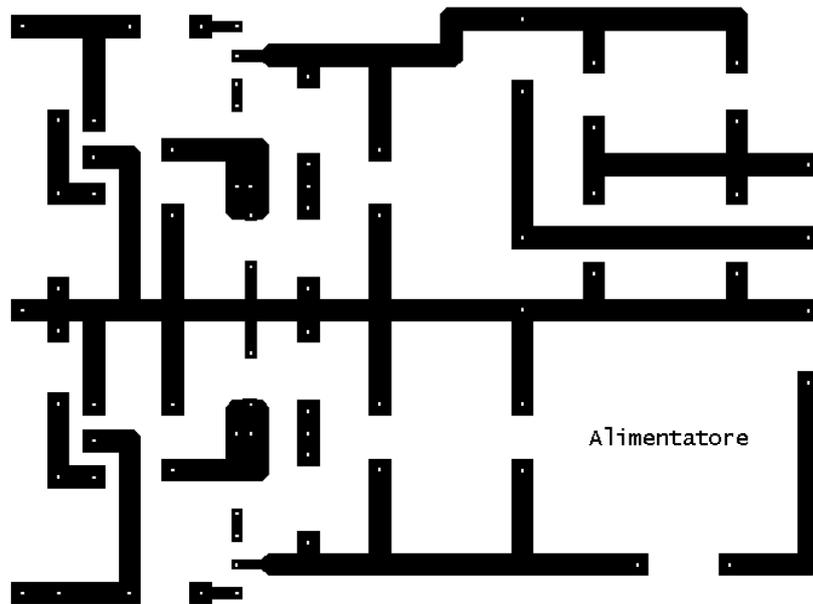
3. *Etching*: at this stage you can immerse the PCB in the ferric chloride solution and wait till all the excess copper has been removed. Warming and agitating the solution accelerates the corrosion process. For the corrosive bath, use a small NON-METALIC tray - large enough for the PCB - and pour the solution into it. The PCB should lie with side that is to be etched face up to avoid its touching the bottom of the container, which would inhibit corrosion. Normally, a few minutes (about ten) are enough to complete the etching; however, the result is checked visually: when all the excess copper has been eliminated, the etching is finished. The ferric chloride can be recovered for reuse (its corrosive power diminishes with each use).

At this point, the board can be thoroughly rinsed in water to eliminate any residue of corrosive solution. The protective paint can be removed with ordinary solvents (e.g. turpentine). After another polishing of its copper tracks the PCB is ready to be drilled. Use a larger drill or chamfering tool to cut a wide chamfer around all the holes on the **ground plane** side. The components can be soldered onto the PCB.

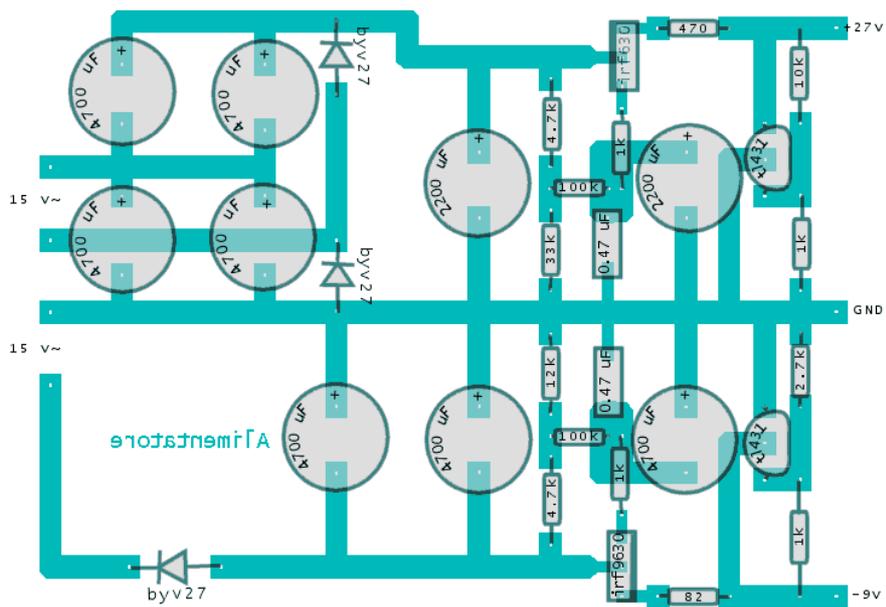
On the following pages, you will find images of the power supply and amplifier PCBs that can be printed and traced onto the copper layer of the boards.

Once the PCB has been fashioned, the components are mounted according to the layouts shown in the figures that accompany the PCB images.

N.B.: the footprint of the capacitors (i.e. the distance between the leads) may vary according to the components selected. Before producing the PCB, it is good to check whether it is necessary to adjust the tracks (even though a certain flexibility has already been allowed for).



The Power Supply PCB



The layout of the power supply components (as seen from the component side)

