The LED Toy

Building Modern Hardware With Modest Means

Werner Almesberger
werner@almesberger.net

October 12, 2006

Contents

Chapter 1

Introduction

The LED Toy is a small electronic circuit, which uses motion to produce 2D images with a single line of flashing LEDs (Light Emitting Diodes). The design is based on an Atmel ATmega48 microcontroller. Program (firmware) and images can be updated over an RS-232C port from a PC.

All the information needed for building and programing this device, including schematics, PCB layout, and the firmware, is freely available and can be downloaded from the project Web site at `http://ledtoy.sourceforge.net/`

This document describes how to build the LED Toy hardware. It does not describe how it works, or how it is used. For all this, please refer to the user's manual [1].

1.1 Why build it ?

First of all, a little box that does not only flash in the dark but even displays little images draws attention not only among technically inclined geeks but also at bars or dance clubs, which makes it an original fashion accessory.

The circuit is very simple, but it incorporates many elements indispensable when building modern electronics, such as a high-resolution double-sided printed circuit board,[1] a microcontroller, and SMT (surface mounted technology, the really tiny chips used in contemporary electronics) components. This makes it an attractive project for exploring this technology, since the amount of work is quite small, the probability of success is high, and it is possible to cover the whole process in detail without having to write an encyclopedia.

Last but not least, since the LED Toy is only of esthetical value, but has no practical use whatsoever, it is unlikely that anyone would consider this circuit not worth building only because it doesn't fit his or her actual needs.

1.2 Who should read this ?

When you flip through the pages of this document, you will quickly notice that even the most mundane matters are treated in almost excruciating detail. The basic assumption is that the reader may be somehow familiar with electronics, but that we can't assume any specific knowledge or experience.

If you are a hobbyist who has already built some non-SMT (surface mounted technology) circuits, you will find most of the material familiar and just looking at the illustrations will probably give you a clear idea of how this is done. If you have not built your own PCBs (printed circuit boards) yet, you may also find the description of the toner transfer method of interest.

If you already know how to build SMT circuits, there should be little new for you in this document, but please have a look at figure 10.3, which shows the best sequence for adding the electro-mechanical components.

If this will be your first attempt to build an electronic circuit, most of the necessary information should be there. You should try to enlist someone with some experience in electronics, to guide you through the process and to assist in case of difficulties. Such a person may also be able to help out with tools. It should be possible to do everything without prior knowledge and without help, but it will be more difficult and take more time.

1. A multilayer board with four or more layers would be even more impressive, but producing that seems to be beyond the reach of low-budget projects.

1.3 Can you do it ?

Even with little prior experience, this circuit can be built by almost everyone with enough perseverance. Basic requirements include the ability to see those small components and their solder joints, a quiet steady hand, and patience and determination.

If you have difficulties seeing the contacts of the components, make sure you have a strong light source and try using a magnifying glass. Working in bright daylight is usually best. If you are near-sighted, you may find it easier to work on small components after removing your glasses or lenses.

If you find it difficult to make sufficiently precise movements, try to keep as much of hand and arm on a solid support, so that your muscles have to lift as little weight as possible. Move only one item at once, e.g., when placing components, make sure the board lies on a flat surface and can be easily held in place. Placing tiny bits with a precision of less than a tenth of a millimeter is unusual work for most people, and it is quite normal to feel uncomfortable after a while. In this case, try alternating the type of movements and take a break.

If you get stuck, try to find someone who has some experience with electronic circuits to look at your work. Even if this person is no expert, you are likely to receive comments and suggestions that will help you to proceed.

The whole device can be built in 3–4 hours if you know exactly what you're doing. Building it for the first time takes about twice as long, provided that you're already familiar with the concepts and tools. If this is your first SMT circuit and your first self-made PCB (printed circuit board), plan on spending about two days on it. If this is your very first exposure to electronics, you'll probably need one or two days more.

You will also need some time to purchase or borrow tools, consumables, and components.

1.4 Why this document was written

There is also a lack of comprehensive material that describes how to build modern circuits with simple tools. Virtually all introductions to circuit-building shy away from the apparent complexity of assembling SMT designs, and while there are many very well-written Web sites describing certain aspects of making even quite advanced circuits, none seems to explain the whole process. Last but not least, the choice of materials and tools also depends to some extent on regional availability, and TO DO: **complete and move**.

1.4.1 About the author

This is usually the place where the author justifies his claim on the reader's time and attention by detailing his exquisite education in the topic matter and the decades of practical experience and prestigious projects that have honed his skills to perfection.

This author has a software background and tinkers with electronics only as a hobby. The LED Toy is his first SMT design and his second project using a microcontroller.[2] Most of the material covered in this document has been gathered from various sources, including a fair amount of trial and error, in the course of about one year. Given that the author could hardly style himself as an expert in electronics, isn't it a little presumptuous to try and write what claims to be a guide for getting started in a field as complex and varied as the production of electronic circuits ?

To tell the truth, it probably is. However, having just gone through the process of figuring out how to accomplish these things, the author feels that this is a good moment to write about them: first of all, because the memory is still fresh, and second, because TO DO: **complete**.

1.5 Document status

This document is still unfinished.

Furthermore, some of the pictures are taken from an earlier version of the LED Toy. TO DO: **explain**

1.6 Legal matters

The unavoidable and tedious. TO DO: **complete**

2. But don't worry too much — it went through a number of iterations, so the worst bugs should be gone by now.

1.6.1 Copyright

When creating a derivative work, you're kindly requested to adhere to the following guidelines:

- If recipients of the derivative work are likely to consider you as a more relevant contact regarding the overall content than any of the other authors, put your name before and, where applicable, in larger print than the names of the other authors. Examples for this include translations, adaptations to other projects, and if you take over maintaining the document for your target audience.

- If your changes depart significantly from the original work or even contradict it, include a summary of your changes in the document, preferably in the "Document revisions" section following this one. Examples include replacement of some technique with a better one, the addition of new component types, or addition or replacement of a large number of figures.

- If you create a compilatory work, identify your role as an editor, place the names of the authors such that they can be easily found, and indicate in which part of the compilation their work appears.

- If the total material copied into your derivative work does not exceed what is contained in a regular printed page of this document, no attribution is required. This does not relieve you from complying with all other licensing terms, in particular adequately notifying the recipients of their rights.

1.6.2 Trademarks

Dremel is a trademark of S-B Power Tool Company, Chicago. In this document, we use the name in lower case (i.e., "dremel") to refer to any device similar in purpose and function to the Dremel Rotary Tool.

Virulana is a trademark of MAPA VIRULANA. In Argentina, "virulana" is used as a synonym for finely-spun steel wool.

Plexiglas is a registered trademarks of Arkema.

Atmel and **AVR** are registered trademarks of Atmel Corporation.

1.7 Document revisions

This is a preliminary version, revisions of which are not yet recorded.

Chapter 2

A quick overview

2.1 Process goals

The circuit building process described in this document should be simple and inexpensive, yet allow the use of state of the art components. More precisely, there are the following goals:

- TQFP / SSOP / 0603 / 0.5mm grid
- all items (except components) available locally
- as little ”wet chemistry” as possible
- no fancy machines required
- no exotic chemicals
- upgradeable

extensions: MLF, 0402, ...

2.2 The circuit board

The printed circuit board (PCB) fulfills two very important tasks: (1) it is the mechanical basis on which all the components of the circuit are placed, and (2) it provides most if not all electrical connections between components.

Producing the circuit board is a fairly involved process, but with a little practice, very good results can be obtained in a short time. In particular, the process described in this chapter allows us to make boards for tiny components down to a pin spacing of 0.5 mm.

If you consider it too tedious to make your own board, there are two alternatives: (1) there are various companies that will, at a price, manufacture boards with a layout you provide. These boards will in general be of better quality than the boards produced with the process described here. Drawbacks of having someone else make the board for you include longer turn-around time (days or even weeks, instead of maybe an hour), higher cost, particularly if your own time is cheap, overhead in converging on the data format and content of the layout, and lack of the feeling of accomplishment you would experience after you've made your own board. (2) you can make more than one board in one pass. Since our circuit is quite small, you can combine several such circuits on a single board, greatly reducing the amount of work and increasing the likelihood of being able to get friends to help you.

The technique we use is called “toner transfer”. It works by printing the layout with a laser printer on a piece of paper and then transferring the toner from the paper to the copper on the board using heat and pressure applied with a clothes iron. The board can then be etched with acid. The acid removes the copper, except for those parts which are protected by the toner.

Finally, there are various simple mechanical tasks, such as cutting the board, and drilling holes into it.

2.2.1 Soldering the components

2.2.2 Building the case

2.2.3 Component availability

In some places, e.g., in Argentina, obtaining SMT components in small quantities and at reasonable prices is very difficult. One reason for this may be that people who normally buy components in small quantities also tend to believe that SMT circuits are impossibly difficult to produce. Therefore, they try to design around the use of such components, creating no demand for them, and the local shops then have little commercial incentive to carry such items.

Components not available locally can be purchased by mail order. The main problem with this is the disproportionally high shipping cost. This cost can be mitigated to some extent by pooling purchases from several people into a single order, and splitting the shipping cost. Group purchases have the additional advantage of making it easier to include a few spare parts in the order.

On the longer term, it may even be possible to convince resellers to offer some of the components used in this project, perhaps even in the form of a kit.

2.3 Workplace considerations

2.3.1 Ventilation

2.3.2 Dust

2.3.3 Noise

2.3.4 Disturbance

2.4 Safety

Many of the materials used and produced when building an electronic circuit can cause irritation or damage when brought in contact with the human body. In particular, eye contact with sprays, liquids, or particles must be avoided.

The following sections describe the hazards involved and provide recommendations for reducing exposure.

Chapter 3

Preparing the circuit board

In this chapter, we prepare the raw board for toner transfer. This chapter is fairly long, because we have to introduce many tools and concepts, but the actual work only takes a few minutes.

Figure 3.1 illustrates the process: we first cut a piece of the size we need from the copper-plated board we bought. Then we clean from oxidation and other stains it with steel wool and alcohol until it is clean and shiny.

3.1 Safety

Many of the materials used and produced when building an electronic circuit can cause irritation or damage when brought in contact with the human body. In particular, eye contact with sprays, liquids, or particles must be avoided.

In this chapter, we will encounter many of the "dangerous" parts of the project. Please read this section carefully. It will allow you to properly assess the risks, some of which you may not be aware of, and help you to avoid accidents.

3.1.1 Cutting and drilling

All tools designed to cut through or drill into hard materials will also do so very efficiently with any body parts they are directed at. Therefore, a few precautions are in order. First of all, keep your fingers at a safe distance from any cutting or drilling tool, and in particular out of the path the tools is traveling, as this is the most likely direction it will take when going astray. Also bear in mind that tools can break, with splinters flying around. It is therefore a good idea to wear safety glasses.

In order to maintain control of the tools, always make sure that the item you are working on is on a solid surface and does not slide around. The only thing moving on your bench should be the tool.

When cutting, use a metal ruler to constrain the movement of the knife and to protect your fingers. If using a ruler is not practical, at least place your fingers such that you cut away from them.

3.1.2 Solvents and sprays

Isopropanol, acetone, acrylic spray, and some flux removers are highly flammable, can cause eye and skin irritation and damage, as well as other types of unpleasantness when inhaled or ingested. Other flux removers, while not flammable, are still toxic. In general, treat these substances as you would treat household cleaners and cosmetic sprays, i.e., avoid inhalation, ingestion, any eye contact, and unnecessary skin contact.

Note that acetone dissolves many types of plastic, so storing it in a plastic bottle of unknown composition may yield surprises, possibly only after a few days. Spilled acetone may in turn act as an irritant and become a fire hazard.

"Compressed air" is in fact not air but an inert gas with relatively benign characteristics. Since this gas can get quite cold (due to evaporation, particularly if the spray can is held upside down when releasing the gas), it can cause frostbite. The precautions outlined above are adequate.

3.1.3 Metal and other particles

When cutting, drilling, and cleaning the board, dust of the board materials, i.e., pertinax or epoxy and copper, is released. Dust production is particularly intense if using a dremel.

Copper causes strong irritation and possibly damage to eyes, nose, and throat, as well as various other

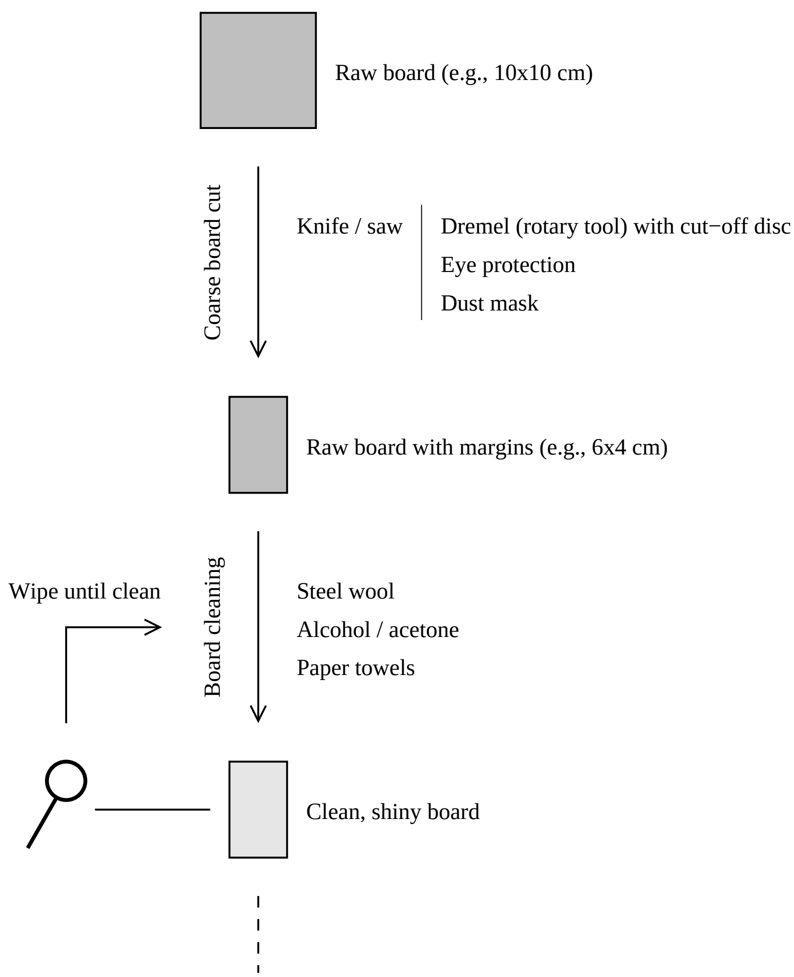

Figure 3.1: Conditioning the copper-plated circuit board.

types of unpleasantness if ingested. Particles produced by manual or low-speed tools have very little momentum and no special precautions are necessary. To avoid ingestion of copper, wash your hands before handling food.

When using a dremel, some additional precautions must be taken: Since the tool will blow fine dust right into your face, you need to wear a face mask and protect your eyes.[1]

You have to wear protective glasses also because the cut-off disc can (and will) break, yielding hard and usually hot fragments flying in arbitrary directions. The disc rotates at about 30'000 revolutions per minute and has a diameter of a bit more than 20 mm. Therefore, the top speed of the fragments should be roughly 100 km/h. You don't want any of this in your eyes.

While scrubbing the board, small amounts of copper and steel dust will be released. While it is unlikely that you will accidentally inhale any significant quantity of it, it will stick to your hands, and you should wash them before touching food.

Particles from pertinax (chemically treated paper) and epoxy (glass) are also at best unpleasant when inhaled or when making eye contact. The precautions against copper will also provide adequate protection from these substances.

3.2 The bare board

A simple printed circuit board consists of a rigid and isolating material, about 1.5 mm thick, which is plated with thin sheets of copper on one or both sides. The isolating material is typically either pertinax, a chemically treated paper, or epoxy reinforced with glass fiber.

Pertinax is cheaper than epoxy, but is less rigid, absorbs moisture, and provides less adhesion to the copper. Lack of adhesion means that, if a solder joint is overheated or is put under mechanical stress, it may separate from the board. Since our circuit is quite small, mechanical properties of the board are of little concern, and pertinax is quite sufficient. The board production process is identical for epoxy, except that the material is harder and therefore slightly more difficult to process with manual tools.

The choice of the board material also affects the geometry of holes. We need holes for through-hole components (see section 10.2.1) and for vias (see section 10.1). Each hole is surrounded by a ring of copper, to which the pin or wire of the component or via is soldered. The ratio of the ring diameter versus the hole diameter determines how much copper surrounds the hole, and thus the size of the surface bonding board and copper. If the adhesion between board and copper is strong, a smaller surface is needed, and the diameter of the copper ring can be reduced for the same hole size. [2] suggests a ratio of 1:2 for epoxy and between 1:2.5 and 1:3 for pertinax. Table 3.1 shows that the ratio in the LED Toy circuit can be as low as 1:2. While this would indicate that we should use epoxy or add more copper, results are satisfactory – within the constraint of this project – with these dimensions even if using pertinax.

Boards are available in a variety of board and copper thicknesses. In principle, anything but the most exotic boards will do. Most shops in Argentina will only have one thickness of boards anyway, relieving you from making a difficult decision.

Copper thickness is usually specified in ounces (per square foot). The most common weight is 1 oz, which corresponds to a thickness of 35 μm.

Boards can have copper on one side (single-sided) or on both sides (double-sided). For our project we need a double-sided board. Some people may tell you that double-sided boards require complex machinery and chemistry to make. While this is true for certain types of double-sided boards, those difficulties are greatly exaggerated, as we shall see.

Industrially produced circuit boards may even have more than two layers of copper. You need such boards for circuits with complex electrical characteristics or a high density of connections. Unfortunately, manufacturing such multi-layer boards is not within easy reach for hobbyists.

3.3 Cutting the board

Bare boards are usually sold in sizes like 10×10 cm^2, 10×15 cm^2, 10×16 cm^2, or larger. For our circuit we need only about 4×6 cm^2. So we first cut a piece of the right size from the original board, as shown in figure 3.2. Note that, if you're combining multiple circuits on one board (see section 4.4.2), you have to adjust the board size.

1. If you think that hiding behind masks and glasses is unmanly, you will quickly change your mind after having tasted a mouthful.

Pin/pad type	Ring diameter	Hole diameter	Ring:hole ratio
Via	65 mil = 1.65 mm	0.5 mm	3.3
Through-hole component	68 mil = 1.73 mm	0.8 mm	2.16
Header	80 mil = 2.03 mm	1 mm	2.03

Table 3.1: The ratio of ring (pad) diameter to hole diameter for holes in the LED Toy circuit.

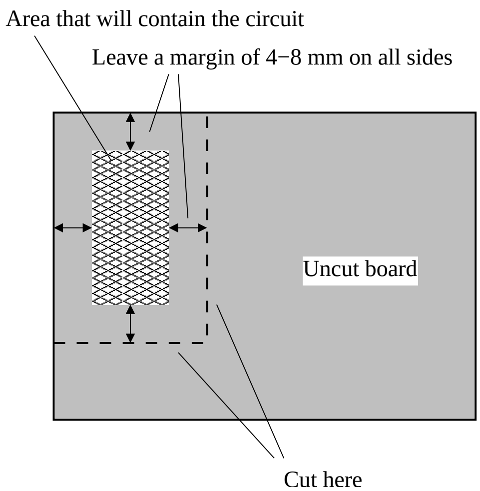

Figure 3.2: Cut a piece from the raw board material, leaving a margin of at least 4 mm around the area that will later be covered by the circuit.

To determine the right size of the piece to cut, print and then measure the layout. It should fit comfortably on the board, with a margin of at least 4–8 mm on each side. Smaller margins will require higher precision in later processing stages and thus should be considered only if you have familiarized yourself with the process. Wider margins waste board material and transfer paper.

If you arrange the cuts such that two or more sides of the raw board coincide with a side of the cut board, only 1–2 cuts are needed per board. It usually helps to mark cut lines with a permanent marker.

3.4 Cutting tools

We can choose among several types of tools for cutting the board:

- A hand saw and a knife. This is an economical choice that only requires moderate skills.

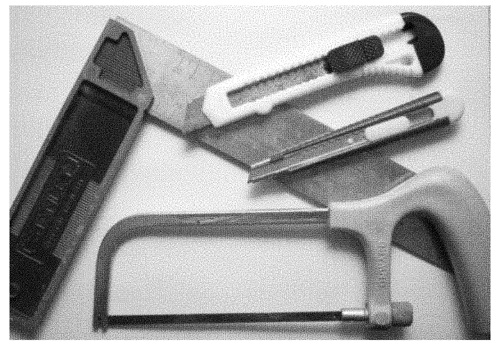

Figure 3.3: Cutting tools from top to bottom: large cutter, small cutter, and a small saw. Below the tools, there is a metal ruler.

- Just a knife. While you avoid needing a saw, using just a knife requires more skill and energy, and it also increases the risk of injury and of damaging the board.
- A dremel (rotary tool) with a cut-off disc. This is by far the most convenient and precise choice, but such a tool is somewhat pricey and produces considerably more dirt than any of the manual tools. Nevertheless, if you are regularly doing fine mechanical work, this is an excellent investment.

You also need a stable surface onto which you can hold the board while cutting it. When choosing your work place, also consider the amount of dust produced, particularly if using a dremel.

3.4.1 Saw and knife

Manual tools offer a fairly quiet way for cutting a board, which helps to maintain a pleasant relationship with neighbours if you decide to start working in the middle of the night. Do yourself a favour and avoid the cheap saws made almost entirely of plastic. They aren't worth even the little money they cost and break within instants. Note that most saw

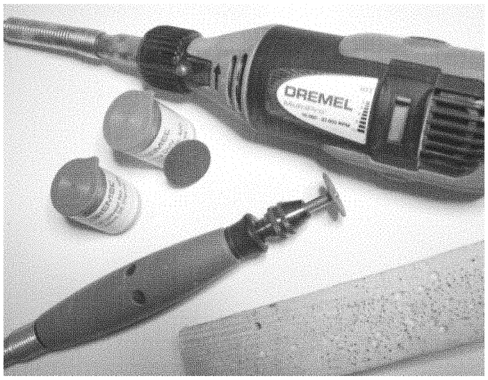

Figure 3.4: More cutting tools: a Dremel rotary tool with flexible axis and a cut-off disc, a set each of regular and "heavy duty" cut-off discs, and a piece of wood to be used as support.

blades, while working well enough on pertinax (i.e., paper), are no match for epoxy (i.e., glass), and dull within centimeters.

If cutting the board with a knife, use a metal ruler for the first cuts. After that, the knife will follow the groove on its own. First, cut through the copper on both sides, then follow the cut six to ten more times (triple this for epoxy) and try to bend the board. If it can be bent without too much resistance, break the pertinax or epoxy by folding it back on itself. If there is still copper connecting the two pieces, either break it by bending it a few times or cut it with a knife.

Remove any copper ridges and any other sharp edges or protruding material with a file or a knife.

Figure 3.3 shows the tools for manual cutting: the saw and the large cutter are good for cutting boards, while the small cutter is convenient for smoothing edges. It is also useful for a range of other small cutting tasks, so it's best to have both types of cutters.

The metal ruler is essential for making long cuts along a straight line, and it also protects your fingers. Place the metal ruler such that the handle hangs off the table, so that its long arm lies flat on the board.

3.4.2 Dremel (rotary tool)

The dremel (rotary tool) is very versatile and precise. Figure 3.4 shows the dremel with a flexible axis. The flexible axis makes handling the dremel a bit easier, but its use is not required. The picture also shows cut-off discs, which are discussed below. Last but not least, the piece of wood serves as a support to avoid cutting or drilling into the table. It can also be used to push the board against the table when using a saw.

When cutting, the main problem is that the cut-off discs break easily if not treated properly. Here are a few suggestions for avoiding this problem:

- Try to use "heavy duty" cut-off discs with a thickness of about 1 mm, instead of the 0.6 mm thin regular discs. The latter are cheaper per disc and let you make finer cuts, but they also break much more often.

- Use the highest speed setting to avoid stalling the disc. If the disc stalls, there is a high risk of breaking it, and you may not have full control over the movements after the disc starts spinning again.

- If the disc stalls, immediately pull it out of the board, going back in the groove you already cut. Then restart from where you left off. The disc will "dig" into the board at any place, a cut doesn't have to start at an edge.

- If the cut goes in the wrong direction, do not try to turn the disc. It will almost certainly break. Instead, pull the disc from the board, then re-insert it with the proper orientation.

- The disc does not have to be fully immersed in the board. If you keep only enough of the disc in the board to reach the other side, you have a minimum of friction and lateral forces. This keeps the disc cool, increases maneuverability, and keeps it from breaking.

If a disc breaks (see figure 3.5), as it sooner or later will, you may find it difficult to remove it from the mandrel, because the screw is quite flat and screwdrivers slide off easily. An effective if somewhat violent approach is to break off all the remaining parts of the disc, and then loosen the screw with pincers. Once the screw is loosened, you can remove it by hand.

Edges can be smoothed by gently sliding the board along the cut-off disc as shown in figure 3.6. Finding the angles and movements (e.g., whether you move the board or the dremel) that work best for you may take a bit of experimenting.

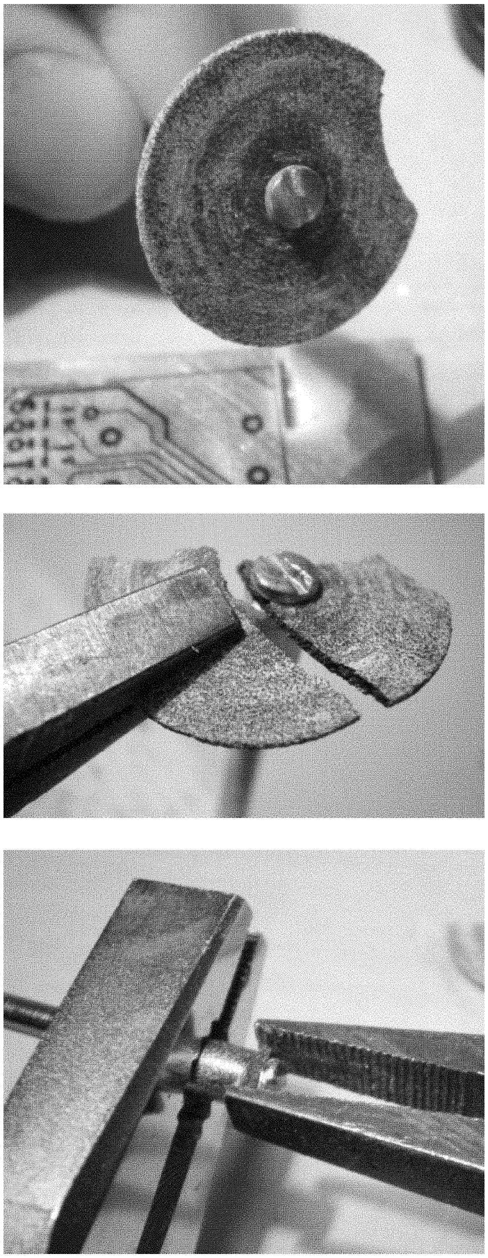

Figure 3.5: To remove a broken cut-off disc (top) from the mandrel, break off the remaining parts of the disc (middle), and loosen the screw by twisting it with pincers (bottom).

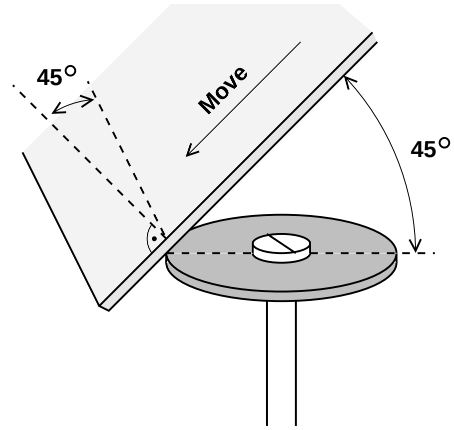

Figure 3.6: Board edges can be smoothed by sliding them along the cut-off disc. This figure shows an example. Experiment to find the best angles and movements.

3.5 Cleaning the board

The next step is to scrub the board with steel wool, to remove fingerprints, oxidation, and other impurities. The steel wool also leaves fine scratches in the copper. The goal of all this is to improve toner adhesion.

Figure 3.7 shows an example of steel wool, and also the alcohol and paper towel we will use later to wipe the board clean. The compressed air is good for blowing away dust, and will be handy through the entire project.

Everybody seems to use a slightly different procedure and different materials. E.g., a popular source for the toner transfer method [3] recommends the use of nylon and acetone instead of steel wool and isopropanol, and to scrub in two directions.

3.5.1 Scrubbing with steel wool

Steel wool can be obtained cheaply from any supermarket. There is coarse steel wool made of thin bands about 0.5 mm wide, and very fine steel wool made of hair-thin strands. The latter scratches the surface only lightly, lowering the risk of making deep scratches. Deep scratches can yield very thin and often invisible breaks in traces.

Scrub in parallel to the long side of the board, until the entire surface is shiny and densely covered with fine parallel scratches. In order to obtain a surface that is evenly treated all the way to the edges,

Figure 3.7: Cleaning tools from left to right: "compressed air", isopropanol, steel wool, and paper towel.

wipe across the board edges. This may cause some steel wool to be torn loose, but a ball of it will still last a long time.

3.5.2 Wiping with alcohol

To remove any remaining traces of dirt and the metal dust left behind by scrubbing, wipe the board with isopropanol (isoprophyl alcohol). Pure isopropanol is available as a spray or in bottles. Either can be used. Isopropanol is also used as a disinfectant for medical purposes, but in that case it is diluted with water, which makes it less suitable for our purposes.

Isopropanol is highly flammable (keep it at a safe distance from soldering irons and other hot objects) and should neither be inhaled, ingested, nor brought in contact with your eyes.

When cleaning the board, place it on a paper towel, so that metal dust wiped off the board does not contaminate the table. Use a generous amount of alcohol for cleaning, and repeat with a new paper towel or a clean surface until no visible stains are left on the paper towel. Then flip the board over, and clean the other side.

During all this, make sure not to touch the copper surfaces, since any grease deposits will affect toner adherence. If you accidentally touch the board, remove the fingerprints with alcohol.

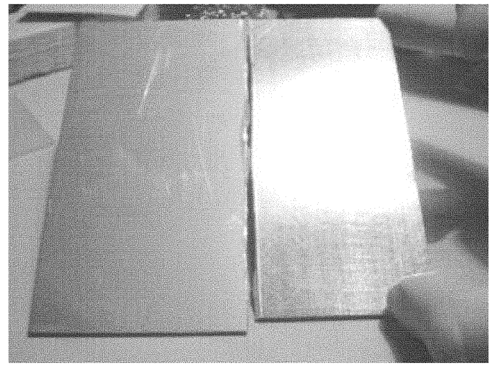

Figure 3.8: Before cleaning (left), oxidation makes the copper look dull. After cleaning (right), the copper is shiny.

When done, the board should look shiny and "metallic", as in figure 3.8. To prevent oxidation, proceed as soon as possible with the toner transfer. If this is not possible, clean the board again with steel wool and alcohol before continuing.

If dust settles on the board, remove it with compressed air. Do not blow on it ! Breath contains moisture and will contaminate the board. Compressed air is an inert gas that is available in spray cans.

Chapter 4

Toner transfer paper

The toner transfer method is a rapid, cost-efficient, and reasonably accurate means for covering the copper of a circuit board with an acid-resistant material. It works by first printing the layout (the image of the traces we want on the board, sometimes also called "artwork") with a laser printer, and then using a clothes iron to transfer the toner from the paper to the copper with heat and pressure. The toner then protects the copper during etching.

Since toner is not easily separated from regular paper, we use plastic-coated paper. Commonly available plastic-coated papers are the paper used to hold sheets of adhesive labels and the "glossy" paper for ink printers.

This chapter describes paper preparation and the printing process, as illustrated in figure 4.1. The actual toner transfer, i.e., the ironing, is covered one chapter later.

Once more, there is a fair amount of theory to cover, so it will take you a lot more time to read this chapter than to carry out the actual work, the latter taking mere minutes.

4.1 Safety and workplace

This part of the process is fairly unproblematic. For the handling of isopropanol and "compressed air", see section 3.1.2.

One thing to keep in mind is that toner transfer is sensitive to dirt and dust. You should therefore avoid touching the surface of the transfer paper, particularly after printing on it. If dust settles on the paper, blow it off with compressed air.

4.2 Transfer paper selection

The choice of transfer paper is crucial for the process. Ironing temperature and duration depend on the paper. Some papers require special treatment, such as surface conditioning, and other papers never yield satisfactory results.

To make sure that the procedure outlined here can be reproduced by people in different regions, we use a mainstream product from a global manufacturer: the "HP Premium Photo Paper, glossy" by Hewlett-Packard, with product code C6039A.[1]

This paper yields good results but is relatively expensive. If looking for alternatives, consider the following points:

- Availability. Since figuring out the optimal way for using a specific type of paper can be quite time-consuming, you want to be sure that this paper will be available on the market for a while. If you intend to build only a very small number of circuits, using a possibly more expensive paper with well-known characteristics is probably still cheaper than experimenting with a set of low-cost papers.
- Surface identification. The plastic-coated surface should be visibly different from the other side or annoying mistakes will happen.
- Temperature tolerance. During the printing process, the paper is briefly heated to about 200 °C. If the plastic melts during this, the paper may get stuck in the printer and may even damage it permanently. In particular, transparencies for ink printers normally melt at low

1. Do not confuse this with the "HP Premium Plus Photo Paper" or the mere "HP Photo Paper". They are different products, with different characteristics.

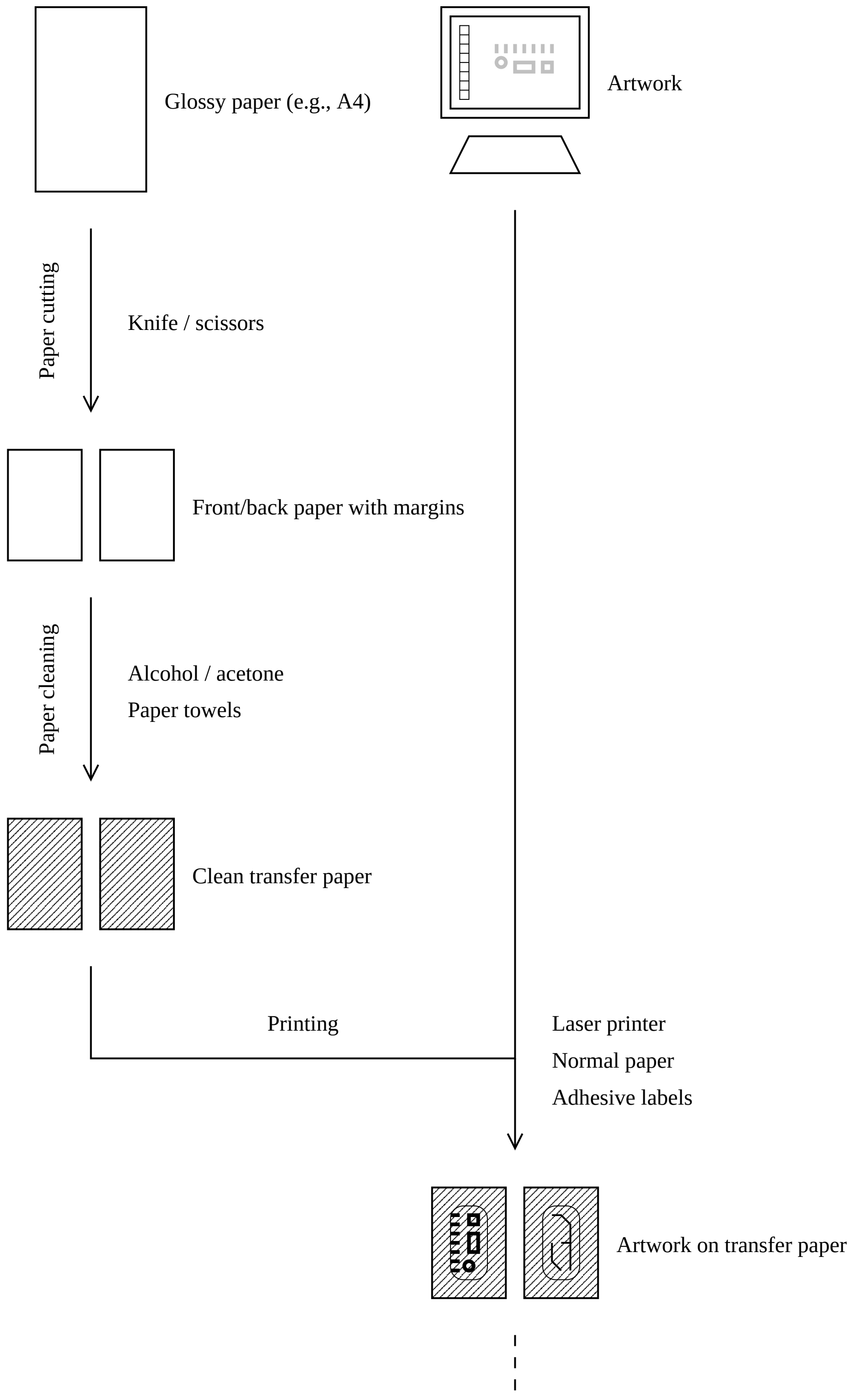

Figure 4.1: Preparing the transfer papers.

- temperatures and jam laser printers in way reminiscent of a high-speed car wreck.
- Integrity. If using paper that normally holds adhesive labels, the structure of the labels may also be imprinted on the paper. Such imprints will cause small defects, typically in the form of fine gaps in the toner, which require manual correction.
- Print quality. The printed image must have sharp edges, there should be no white spots inside traces, and white surfaces should be free from toner particles.
- Clean separation. Some papers stick very strongly to the copper after ironing and may leave small flakes behind, whose removal is messy and time-consuming.
- Transfer quality. The most important criterion is obviously the correct transfer of the toner from paper to copper. The toner must completely cover all traces, without gaps, dents, or holes. Edges must be sharp and the toner must not "leak" outside traces or pads. In particular, separate traces must remain clearly separated.

Papers specifically optimized for toner transfer are commercially available. They probably offer some advantages over improvised solutions such as the one described in this document, but availability and price may make their use unattractive.

In any case, a fair amount of experimenting is required until good results can be obtained. Expect to need several tries until producing an board of acceptable quality. Fortunately, most problems can be identified relatively early, and only the pieces of transfer paper are lost in a failed attempt, while the board itself can be cleaned and reused almost indefinitely.

4.3 Paper preparation

This section describes the few simple preparation steps the transfer paper needs. Like with the circuit board, the principal tasks are cutting and cleaning.

4.3.1 Cutting the paper

The transfer paper usually comes in the form of sheets of roughly A4 or US legal size. From this,

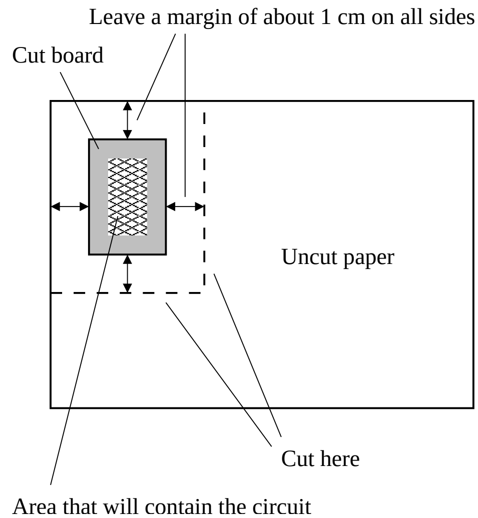

Figure 4.2: Cut the transfer paper such that there is a margin of about 1 cm on each side of the cut board.

we need to cut two pieces, one for the front and one for the rear side of the board. These pieces must be large enough to cover the board we have prepared in section 3.2, with a margin of about 10 mm on each side, as shown in figure 4.2.

Choosing a smaller margin complicates stapling of the two pieces of paper. If the margin is below 3-4 mm, it is no longer possible to make a pocket into which the board is placed. A wider margin only wastes transfer paper.

4.3.2 Cleaning the paper

The surface of the transfer paper may be contaminated with fingerprints or glue residues (in the case of paper holding adhesive labels), or it may be coated with a gelatinous material that interferes with toner transfer. The latter is the case for the "HP Premium Photo Paper" we use as a reference.

Figure 4.3 shows the apparent structure of the paper: below the gelatinous substance is a film of the soft plastic that we want to print on. This film is supported by a thick layer of regular paper. The rear side is sealed with a film of more rigid plastic. When heated, this plastic becomes sticky, so we need to take special precautions when ironing.

Dirt and part of the gelatinous cover can be removed by cleaning the paper with isopropanol or acetone. To do so, place the cut paper on a paper

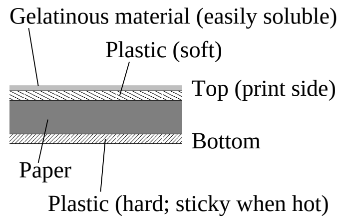

Figure 4.3: The "HP Premium Photo Paper" consists of (at least) four layers of material with distinct properties. The gelatinous material is soluble in water and in alcohol. The figure is not to scale and in the real paper, all the layers are white.

towel, sprinkle alcohol on it, and gently wipe the paper clean with another paper towel. The paper towel underneath the paper prevents us from picking up dirt from the table and carrying it onto the transfer paper.

4.4 Printing

The layout of the board is part of the LED Toy "hardware" package that can be downloaded from the project Web site at `http://ledtoy.sourceforge.net`. The layout containing the main circuit board and the serial interface is called `lta24.brd`. It can be viewed, printed, and edited with "Pcbnew" from the "Kicad" EDA (Electronic Design Automation) system. [?]

To print the layout, start Kicad, select `lta24.pro` under "Projects/Open Project Descr", then either click on the "Pcbnew" icon or double-click on `lta24.brd`. Now, the layout appears on the screen. Click on the "Plot" icon, check "Plot Mirror" if necessary (see below), then click on "Plot".

This generates the following files:

- `lta24-Component.ps`
  The front or component side of the layout, which contains most of the traces. It is shown on top of figure 4.4. This printout is used for toner transfer, in the manner described below.

- `lta24-Copper.ps`
  The back or "solder" side of the layout, which contains some additional traces. It can be see at the bottom of figure 4.4. Also this is used for toner transfer.

4.4.1 Silk

In industrial PCB production, it is common to print information useful during assembly, maintenance, and use of the product on one or both sides of the board, in the so-called "silkscreen"[2]

The "PCB" program also produces a silk screen layer, containing component outlines and annotations. Since this information is already included in the assembly, and since adding the silk layer complicates the board production, the author generally does not suggest to print the front and back "silk" on home-made boards.

While it is possible with the toner transfer method to copy the "silk" after the board has been etched and cleaned from the toner used as etch resist, the result is often slightly mis-aligned and scratches easily.

However, this is ultimately a matter of personal choice, and a properly printed silk layer certainly improves the appearance of the board. If printing a silk screen layer, you may want to consider coating the board (section 14.3), to protect the toner from getting scratched.

Note that isopropanol is a weak solvent for toner, and that flux remover may also remove toner. Therefore, a silk layer can only be applied with the toner transfer method if flux removal is either unnecessary (e.g., when using a so-called "no clean" flux), or if the solvents used for flux removal leave the toner intact.

4.4.2 Layout selection

Several boards can be produced (printed and ironed, often also etched) at once if their layouts are combined and printed together. To do so, open the layout, hold down the "Shift" key, make a block round the board with the mouse, then release the mouse button and place the copy of the board next to it. Be sure not to save the result over the original file.

Remember to leave enough space between the individual boards for separating them before or after etching, see section 8.1. The minimum gap depends on the tools. 2–3 mm are usually sufficient.

The size of the combined board is limited by the size of the raw PCB material and of the size of the etching tray. Furthermore, the larger the board, the more likely flaws will develop during ironing.

2. The name originates from the "screen-printing" or "silkscreening" printmaking technique.

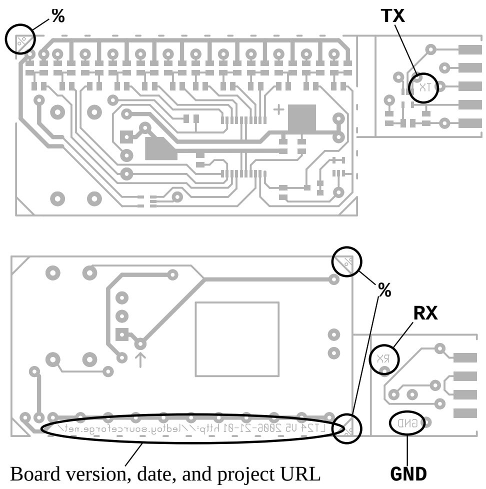

Figure 4.4: The layout is correctly oriented if the text on the printouts for both sides is mirrored.

4.4.3 Layout orientation

For toner transfer, both sides of the layout must be printed such that they appear *mirrored.* The correct orientation can be verified by looking at text that is part of the layout. Figure 4.4 shows places where text can be found in the layout.

When using “Pcbnew”, one can choose to “mirror” under “Plot”. If mirroring is not enabled, the front side is not mirrored, but the back side is. With mirroring enabled, the front side is mirrored, but not the back side. Therefore, the front side has to be printed with mirroring enabled, while it must be turned off for printing the back side.

4.4.4 The transport sheet

The transfer paper is expensive and we want to use as small a piece of it as possible. Unfortunately, when trying to feed small pieces of paper to the printer, these tend to jam or rotate. Therefore, we attach the transfer paper to a regular sheet of paper, which we shall call the transport sheet.

In order to properly place the transfer paper, we need to know exactly where the layout will be printed. While this will be in general “the center” of the page, some printers have very peculiar ideas of where this center may be located. Therefore, we print the layout on the transport sheet, then place the transfer paper exactly on top of the printed layout, and finally insert the two papers to print on the

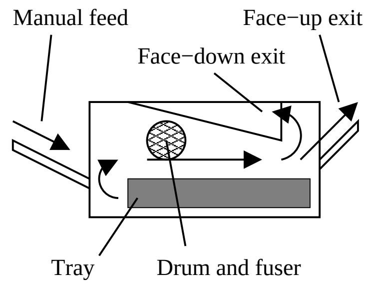

Figure 4.5: Most laser printers offer the choice between several paper paths. Using a straight path avoids complications with the toner transfer paper.

transfer paper.

Before starting to print, make sure to select a reasonably straight paper path, to reduce the risk of the transfer paper separating from the transport sheet and jamming the printer. Many printers offer the choice to pick up paper from a tray or from a manual feed. The latter typically uses a simpler paper path. Furthermore, many printers have a top (“face-down”, regular order) and a rear (“face-up”, reverse order) paper exit. Using the latter avoids curling the paper around one additional set of rollers. Figure **??** illustrates the paper path options commonly found in printers.

Printers with a duplexer unit subject the paper to particularly torturous handling. Fortunately, they are generally “intelligent” enough to avoid engaging the duplexer if only one side of the paper is printed.

Make sure to mark the transport paper before printing, so that you can insert it for the second pass with the correct orientation. This also helps to verify that the printer indeed prints on the side you expect it to.

4.4.5 Printing on the transfer paper

Place the transfer paper on the transport sheet such that the layout is exactly under the center of the transfer paper. If the layout is difficult to see, hold both papers against a light source.

Then attach the transfer paper to the transport sheet. Paper labels for laser printers work best for this purpose. Do not use staples or any other type of metal or other hard material. Also avoid adhesive plastic tape, since it may melt in the printer, staining and possibly damaging it.

Place the stickers such that the transfer paper

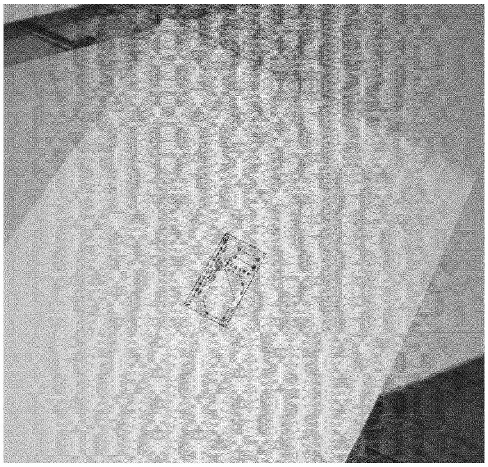

Figure 4.6: The transfer paper is attached to the transport paper by a small adhesive label and then printed on.

cannot rotate or flip over. Also make sure that they do not get near the layout or even cover it.

Then insert the transfer sheet in the same paper supply and with the same orientation as before, and print the layout again. If your printer has an "economy" mode that uses less toner, disable it now. The result should look similar to what is shown in figure 4.6. When done, repeat all this with the layout for the other side of the board.

4.4.6 Quality control

After printing, examine the layout for flaws. In general, if using the right paper, not much can go wrong in the printing process, but you may encounter some difficulties during the first tries. If trying a new brand of paper, print quality may be an issue.

If the paper has shifted during printing, or if part of the layout has been printed outside the transfer paper or on a label used to attach it to the transfer sheet, it cannot be salvaged.

TO DO: **complete** If the toner is very weak If there are occasional thin interruptions in the TO DO: **picture: perfect** TO DO: **picture: blurred** TO DO: **picture: breaks/holes** orientation

Chapter 5

The transfer

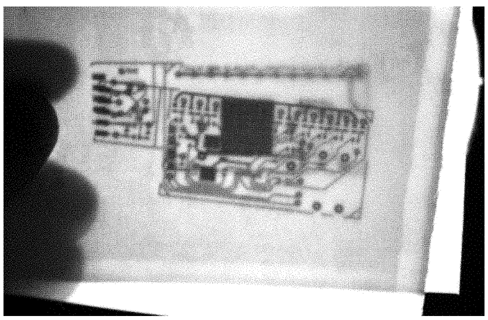

Figure 5.2: Bring both sheets of transfer paper into alignment while holding them in front of a light source.

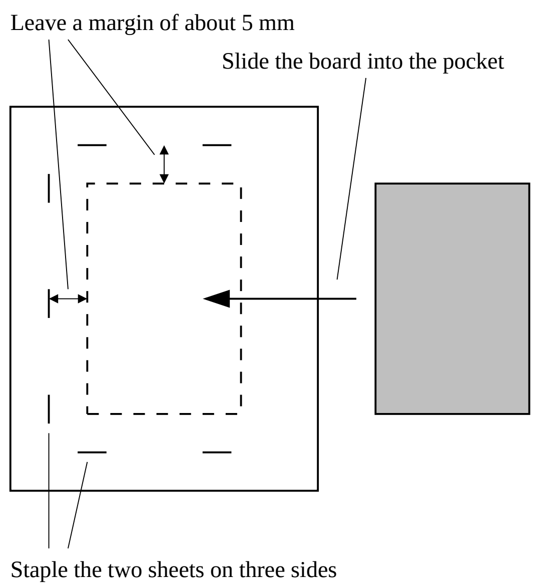

Figure 5.3: Staple the two sheets such that they form a pocket into which we slide the board.

We now come to the fun part of the toner transfer process, the creative abuse of a clothes iron. Figure 5.1 outlines the steps in this phase: we combine the transfer paper and the board prepared in the last two chapters into a single “sandwich”, which we then iron, so that the toner becomes liquid and sticks to the copper.

5.1 Safety

The only thing to keep in mind here is that not only the iron gets hot (treat it with the usual precautions, in particular turning it off after use), but also the board gets a lot hotter than you might expect.

5.2 Aligning both sides

Contrary to popular belief, producing a double-sided board is hardly any more involved than producing a single-sided one. The critical part is to properly align both sides. To do so, bring the two sheets

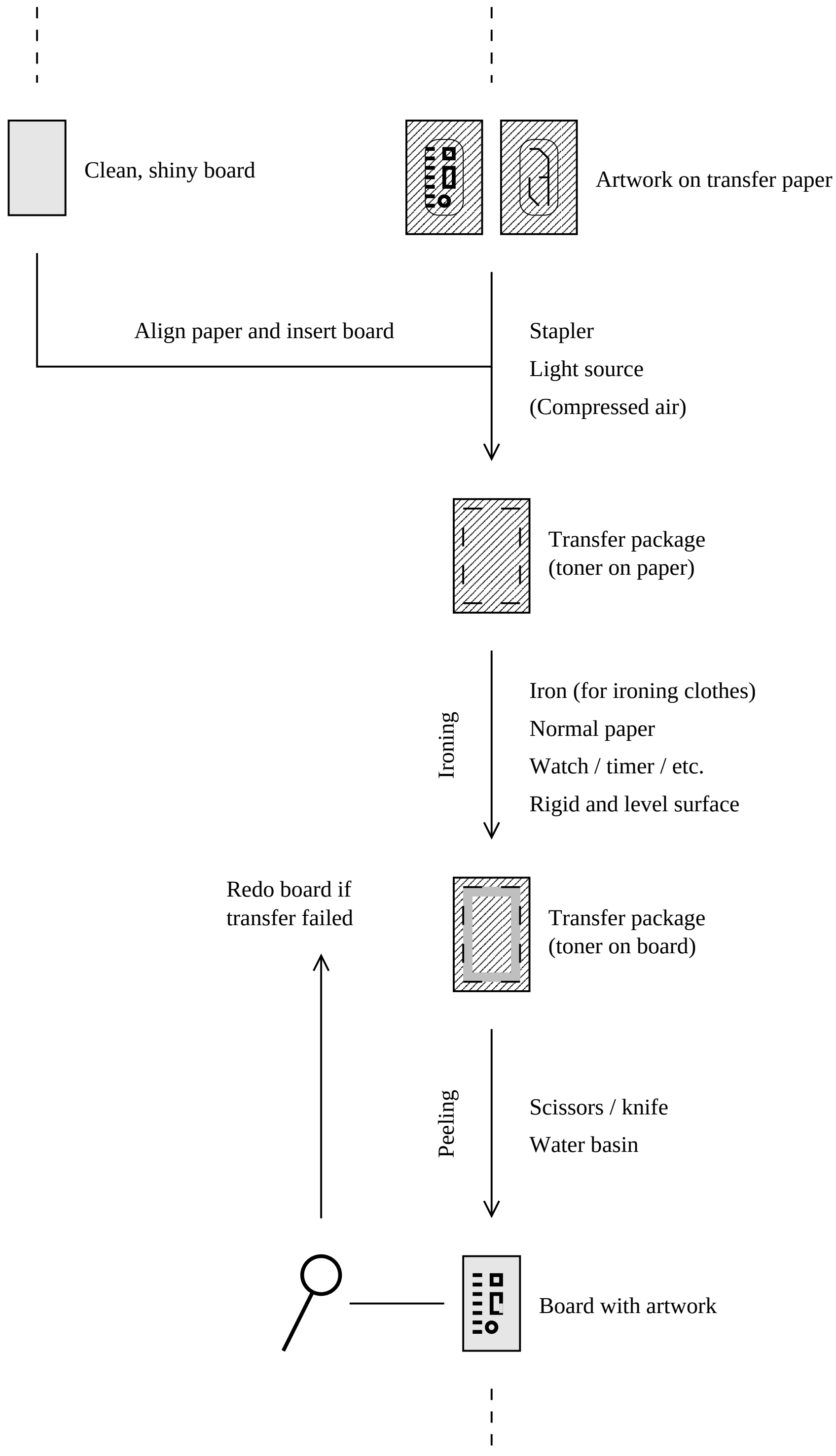

Figure 5.1: Transferring the toner from the transfer paper to the circuit board.

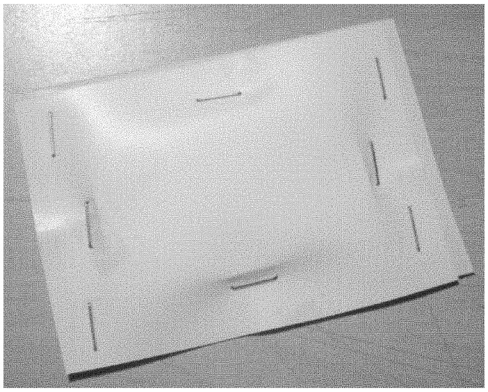

Figure 5.4: The transfer package. The board is fixed between the two transfer sheets with staples.

Figure 5.5: Ironing tools, from left to right: clothes iron with a plane surface, a sheet of paper to protect the iron, and a small stapling machine.

of transfer paper together such that the printed-on sides face each other.

Then hold the papers against a light source (see figure 5.2) and gently shift them until they precisely overlap. This is also a good opportunity for checking that the two sides really match. Staple the two sheets together on three sides such that they form a pocket, as shown in figure 5.3. Use small staples, so that they will not scratch the iron. Remember that the board has to fit into the pocket, so it's better to make it too large than too small.

Obtaining exact alignment can sometimes be difficult, particularly if the paper surfaces are somewhat rubbery and don't slide on each other freely. In such cases, focus on one corner and bring the rest only into rough alignment. After fixing this corner, you can bend the papers a little to bring also the rest into correct alignment.

Remove any dust that may have settled on the board with compressed air. Now insert the board into the pocket and staple closed the remaining side. Add more staples to keep the board from moving inside the pocket. The resulting package should look like the one in figure 5.4.

You should proceed with ironing as soon as possible, as the board can still oxidize while in the pocket.

5.3 Ironing

Ironing requires the application of the right temperatur and pressure in the right pattern over the right amount of time. This usually takes a bit of experimenting to get right, so expect two or three failed attempts before success.

Each attempt will destroy the transfer paper, but the board can be cleaned and reused.

5.3.1 Choosing the iron

All we need the iron to provide are constant pressure and heat. This means that we can confidently pick the cheapest iron we can find, as long as it has a level surface without holes. In particular, neither the capability to emit steam nor any mechanism to sprinkle water will be of any use. Quite to the contrary, humidity is best avoided during this type of ironing. Figure 5.5 shows a simple and cheap iron.

Note that there is a risk of plastic finding its way from the transfer package to the iron, making it dirty and sticky. The staples on the transfer package will eventually leave scratches on the surface of the iron. Neither of this is very welcome if the iron is ever to be used on clothes again. It is therefore a good idea to purchase an iron specifically for "technical" use.

5.3.2 Table surface

The ironing should be done on a hard surface that resists heat. Stone or metal are ideal. Wood is also a possibility, but keep in mind that the heat may discolor and damage the wood. That table or similar should be able to bear your full weight, when pressing down on the iron.

What will not work are any of the supports used for ironing clothes, since they are far too soft to allow proper pressure to be applied.

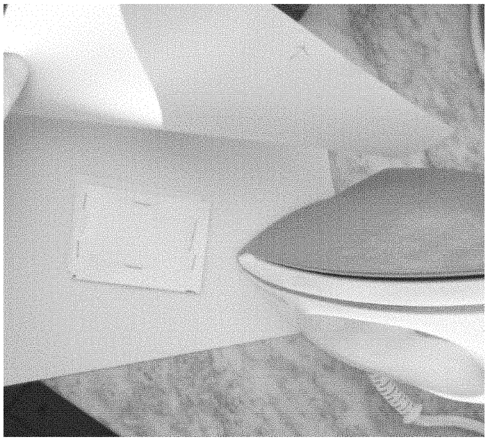

Figure 5.6: Placing the transfer package inside a sheet of paper prevents sticky hot plastic to stain the iron and to make it stick.

5.3.3 Cover paper

The plastic coating on the back side of the transfer paper will melt and stick to the iron. Furthermore, since the two sheets of transfer paper will not overlap perfectly, part of the front sides will face the iron, and stick to it even more.

To prevent all this, fold a sheet of regular paper and put it around the transfer package, as is shown in figure 5.6. Paper towels tear easily and are therefore less suitable for this use.

5.3.4 Temperature

The right temperature depends on the type of transfer paper we use. In general, ink printer paper will need a high temperature, while papers with a high content of plastic work better at lower temperatures. On the iron, the highest temperature is normally "linen". For a lower temperature, the "wool" setting can be used.

If trying a new paper, it is best to start with typical values, and then make adjustments based on the results. Note that many types of paper never yield any acceptable results.

Since we will need a fairly constant temperature, it is a good idea to give the iron some extra time to

5.3.5 Pressure and duration

The heat transferred depends not only on the temperature of the iron, but also on the time during which heat is applied, and the pressure on the iron.

In general, the iron should be pushed hard, but without shifting or tearing the transfer paper. The following movement pattern yields good results with the HP 6039A paper:

- Place the iron so that it covers the entire board, and apply pressure for 20 seconds. The carrier material has now molten, and will stick to board and covering paper.
- Rotate the board by 180°, and apply moderate pressure for another 10 seconds.
- Hold the covering paper, and move the iron along the edges of the board, until they become visible on the covering paper, for 20–30 seconds.
- Make ironing movements on the board surface, alternating the direction and the orientation of the iron, and occasionally applying pressure with the edges of the iron, for about 20–30 seconds.

Then turn the board around, and repeat with the other side. The other side will already have received some heat, so it may need marginally less ironing.

When done, either wait a few minutes until the transfer package has cooled down, or throw it into cold water.

5.4 Paper removal

TO DO: **removal: soak 1/2h in hot water**

5.5 Toner inspection and repair

TO DO: **pictures of failures**
- blurred - uncovered areas - pinholes - paths

5.5.1 No transfer

5.5.2 Blur

5.5.3 Pinholes

5.6 Starting over

Figure 5.7: Toner touch-up and removal: bottle of acetone, permanent marker, and an etch-resist pen.

Chapter 6

Etching

6.1

6.1 Safety

Among the substances whose use is described in this document, ferric chloride acid is the most dangerous one and it has to be treated with respect. It is corrosive and any contact with eyes must be avoided. Likewise, avoid ingestion and skin contact. Small amounts of chloride gas are released during etching. Avoid eye contact and inhalation of this gas and ensure good ventilation.

Wear latex gloves when handling acid and avoid situations where acid could spill, e.g., as the result of items falling into the acid, or of an open container dropping to the floor. The use of safety goggles is recommended.

Spills of ferric chloride acid on any material, including metals, ceramics, and clothes, will leave permanent and clearly visible stains. Remove any spilled acid immediately with water and soap. To protect clothes and skin, a plastic poncho should be worn.

Used acid is still corrosive and toxic, may attack piping, and contains copper, which itself is toxic (see section 3.1.3). In particular, copper is a fish poison. For all these reasons, proper disposal of used acid requires that the acid is neutralized and that the copper is removed, e.g., through sedimentation.

Store acid, new or used, such that it is out of the reach of creatures who may not fully understand its dangers. Also make sure its container cannot be mistaken for anything harmless.

6.2 The acid bath

6.3 Etch capacity

Perhaps you are wondering how many boards you can make with an acid bath before acid weakens and needs to be replaced. This depends on a number of parameters, which we discuss below.

The etch capacity is the amount of copper a certain quantity of acid can etch before losing its strength. Common ferric chloride etching solutions are rated at 15–19 ounces of copper per gallon of solution. We use the average of 17 oz/gallon, or 127 g/l.

We can use this to calculate how many boards we can etch before we have to replace the acid. The usual quantity of copper per side of a PCB is 1 ounce per square foot, or 305 g/m^2. We will remove almost all of this copper, retaining only an estimated 10% of the entire board (including borders that will be cut off later) for traces.[1] Since we use double-sided boards, we need to double this number, so we obtain 305 g/m$^2 \cdot 90\% \cdot 2 \approx 550$ g/m^2.

Next, we need to know how much acid there is per surface of the board. This depends on two factors: (1) how deep we make the bath, and (2) how large the etching basin is in relation to the board surface. We assume a board size of 6 cm $\cdot$ 4 cm = 24 cm^2. The basin shown in figure **??** has a bottom surface of roughly 10.5 cm $\cdot$ 10.5 cm $\approx$ 110 cm^2. Thus, for each unit of board surface, we get about 4.5 units of basin surface.

The average board will be 2 mm thick. In order

1. This figure is for the LED Toy. In circuits where large ground areas are placed between traces, much less copper is removed. If making such a board, adjust the numbers accordingly.

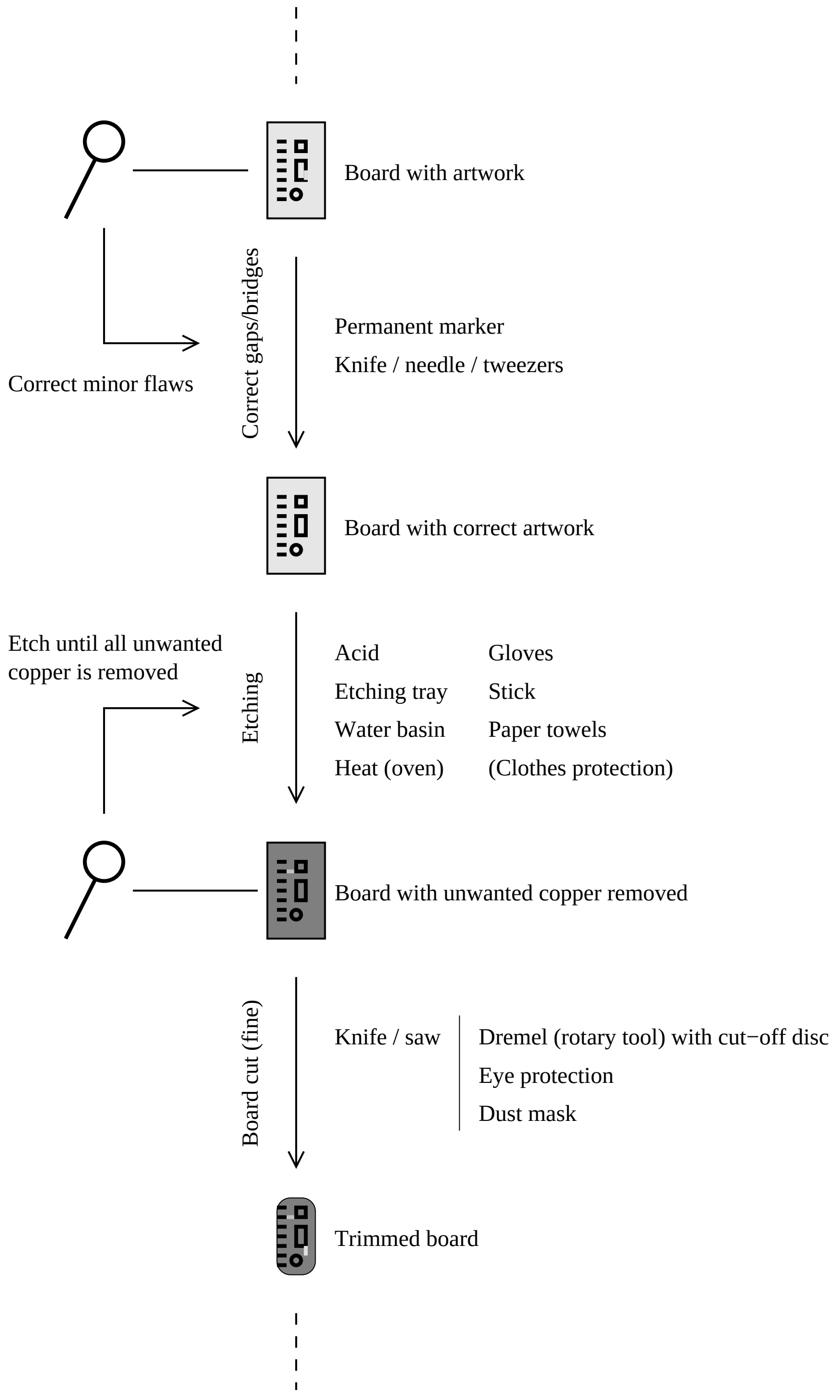

Figure 6.1: Etching the board.

to have the board well covered with acid, we fill the basing to a depth of at least 3 mm. (That's 33 cm^3, so the common half-liter bottle of etchant solution will yield up to 15 baths.)

We can now put the items together: for each square meter of board, we have $4.5 \cdot 3$ mm = 13.5 mm of acid, or 13.5 l/m^2. This quantity of acid etches up to $13.5 \text{ l/m}^2 \cdot 127 \text{ g/l} \approx 1.7 \text{ kg/m}^2$ of copper.

Since a square meter of our board contains 550 g of copper, we can thus etch about three such boards before we need to replace the acid. In practice, we can probably squeeze out a fourth board, but the acid will be weak, and etching will be considerably slower than for the first boards.

Chapter 7

Drilling

The circuit contains a small number of through-hole components and a few vias (connections between both sides of the board), for which holes need to be drilled into the board.

After that, we remove the toner and correct any flaws that may surface at this point. Note that the toner is left on the board during the drilling, because it adds to the height of the sides of holes and thus helps the drill to center itself in a hole.

The general work flow is depicted in figure 7.1.

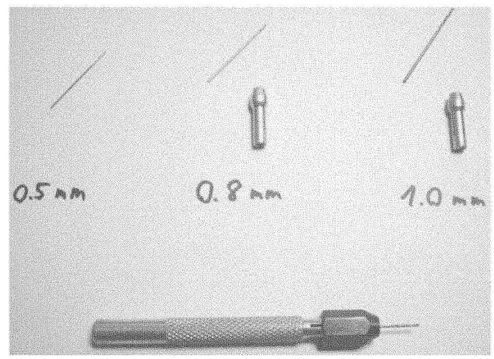

Figure 7.2: Drilling tools: three size of drill bits, collets for two of them, and a manual drill.

7.1 Drilling

Besides the obvious size advantage, there are few things that illustrate the advantage of surface-mounted over through-hole technology more impressively than the amount of work spent on drilling holes: the LED Toy only needs about 30 holes, most of them easy ones for vias. Doing the same circuit in through-hole technology would require at least 130 holes.

7.1.1 Drill bit sizes

The most common hole diameter is 0.75–0.8 mm. This is suitable for most wires and through-hole components. Components that are exposed to mechanical stress, such as connectors and switches, have slightly thicker pins, and need a hole diameter of 1 mm.

For vias, an even smaller hole size, e.g., 0.5 mm, offers the advantage that the wires will fit so tightly that they will stay in place without bending them. This simplifies cutting. See section **??** for details.

The use of a correctly sized drill bit is important. In particular, drilling a 0.8 mm hole with a 1 mm drill will remove much of the copper surrounding the hole, which complicates soldering and increases the risk that the copper separates from the board under mechanical stress. If in doubt, measure the bit with a caliper before using it.

Figure 7.2 shows three different drill sizes and a caliper at the top.

7.1.2 Drilling machines

Since the drill bits are fragile and holes must be drilled with tolerances well below one millimeter, regular do-it-yourself drills (the ones used to make holes in walls) are unsuitable for our purposes.

TO DO: **tools-drill: rearrange according to text below**

The instrument of choice is the dremel (rotary tool). It is designed for this type of precision work and can handle small drill bits. Small drill bits require either a special collet or an adjustable mandril. Both are shown at the bottom left of figure 7.2. If you have a reasonably steady hand, a drill

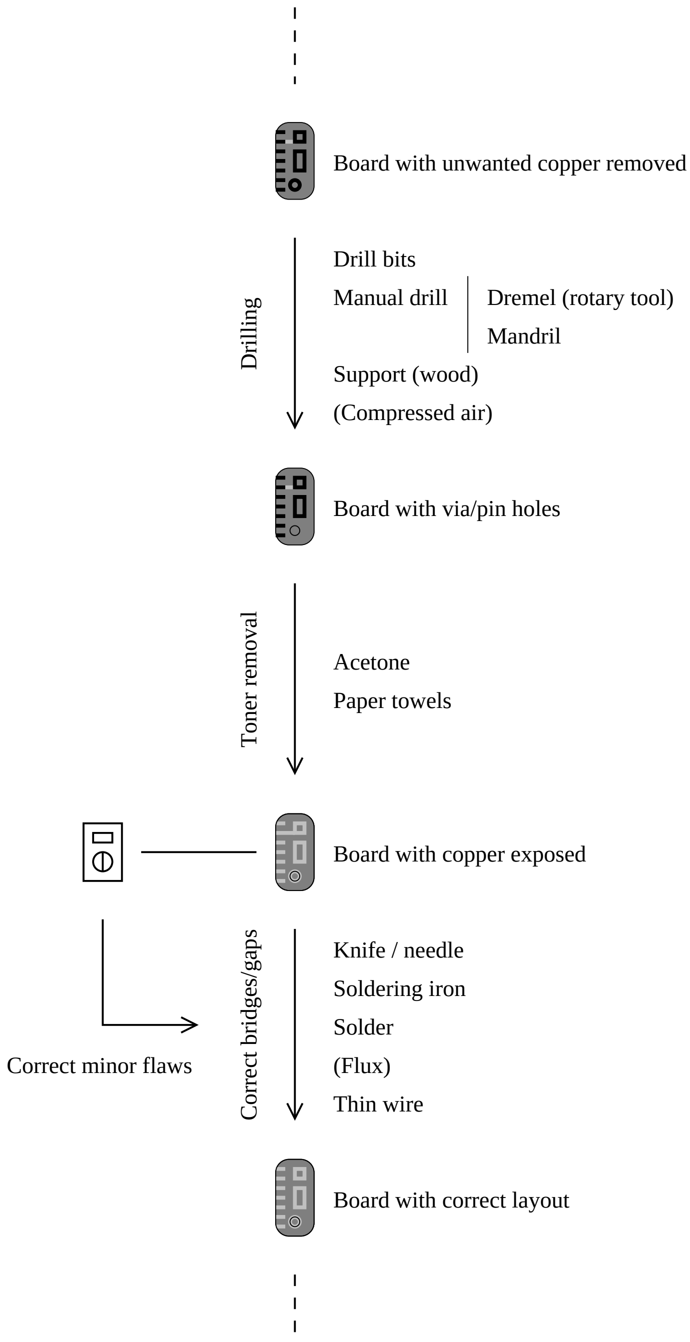

Figure 7.1: Drilling holes for vias and through-hole components.

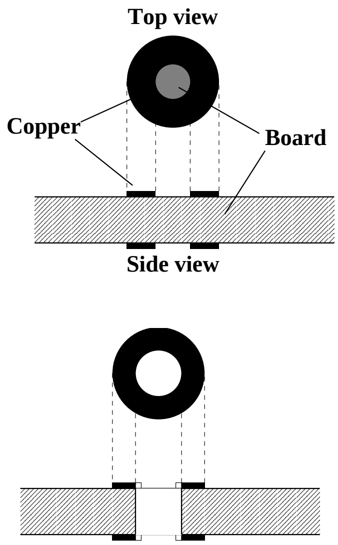

Figure 7.3: The constellation of board and copper, before (above) and after (below) drilling a hole.

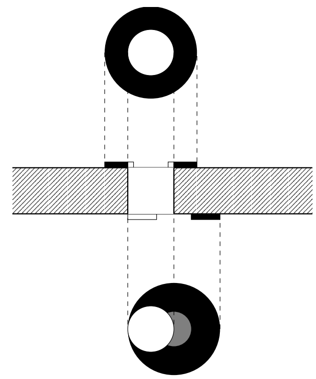

Figure 7.4: Offset can cause very ugly holes. In the bottom hole of this example more than 10% of the copper is lost.

stand is not needed and may actually yield worse results. If using pertinax, the drill can be operated at a low speed setting, so even nocturnal drilling is an option that does not involve waking the neighbourhood. Epoxy requires slightly higher speeds.

Given that the number of holes is low, the use of a manual drill is an elegant and cost-effective alternative to a dremel. This type if drill is shown at the bottom right of figure 7.2. A disadvantage of using a manual drill is that it is difficult to drill diagonally, so there is slightly less tolerance to the misalignment discussed in the next section.

7.1.3 Hole misalignment

Figure 7.3 shows what a hole looks like from the top, and what it would look like from the side if we cut the board across its center: before drilling, there is just a ring of toner-plated copper on each side of the board. After drilling, the hole goes exactly through the centers of both of these rings.

Note that the hole is slightly wider than the inner diameter of the ring, so that a bit of copper is removed in the process. This is done in order to allow for some imprecision in the drilling.

Unfortunately, our production method does not guarantee perfection. First of all, the two sheets of transfer paper may have been imperfectly aligned or have gotten bent in the process. Then, they may have shifted with respect to each other when pushing the board between them. Last but not least, the sheets may also have shifted during ironing. Whatever the cause, there is usually some small offset between the two sides.

Figure 7.4 shows what can happen when we drill through the center of the copper ring on one side if the other side is not correctly aligned. Worse yet, we may even drill through a trace connecting to this pin. The best way to determine direction and magnitude of the offset is to find a few holes in areas of the board that are not overly densely populated, and to drill through them. Note that the offset may change across the board, e.g., if the two sheets have rotated with respect to each other.

If the offset is large enough to merit special attention (e.g., > 0.2 mm), one can distribute it evenly on both sides by drilling slightly off-center, as shown in figure 7.5. If doing this, check that your correction does indeed go in the right direction and does not actually *in*crease the deviation.

If using a dremel, we can try to compensate for the offset even more accurately by drilling at a slight angle, as shown in figure 7.6. This is only recommended for vias and components with a small number of pins. Holes for connectors and headers should always be perpendicular to the board.

7.1.4 Irregularly shaped holes

Some components, typically switches or connectors, have pins with a profile that differs considerably from a circle or a square and need an elongated hole.

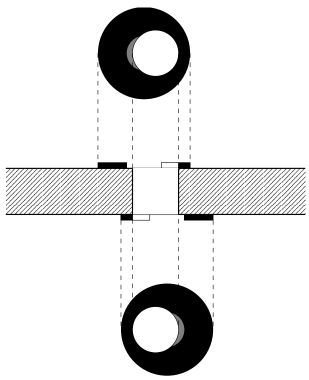

Figure 7.5: For holes that can only be drilled vertically, try to distribute the error evenly. In this example, each ring loses about 3% of the copper it would have with a perfectly centered hole.

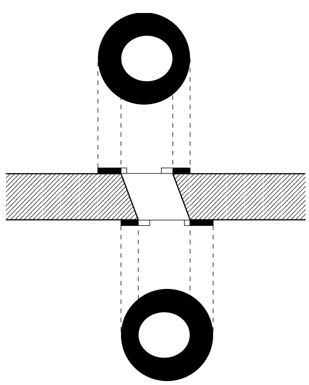

Figure 7.6: An offset between both sides can be compensated by drilling diagonally. In the example shown, each ring has about 4% less copper than with a perfect hole. In reality, the board is thicker in relation to the hole diameter, and the loss would be only about 2% per side.

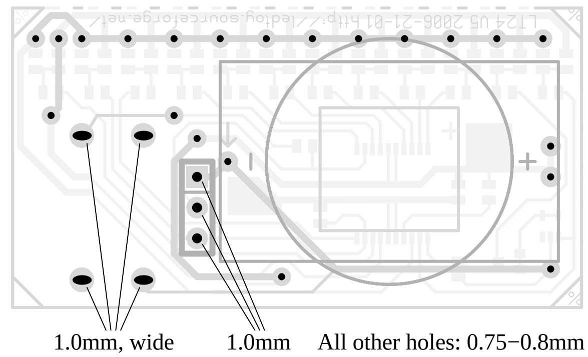

Figure 7.7: ...

Such holes can be produced with a dremel by first making the central hole, then drilling into that hole at a relatively flat angle (up to perhaps 45 °), and finally inserting the drill vertically again and gently pushing it against the sides.

If using a manual drill, the central hole can be enlarged by stabbing at its edges with a small screwdriver.

7.1.5 Hole locations and diameters

??

Chapter 8

Finishing the board

8.1 Trimming the board

Our board is still considerably larger than the circuit we need in the end. We could proceed now with etching the board in its current state and remove the excess material later, or do the trimming before the etching.

8.2 Trim now or later ?

The disadvantages of doing the trimming later are that the etching will require a larger tray, simply because the board is larger, and that acid will be wasted on removing copper we might as well cut off. As an example, the main circuit of the LED Toy covers an area of 25×47 mm^2 or 1175 mm^2, while a board with a margin of 5 mm on each side covers 35×57 mm^2 or 1995 mm^2, which is almost twice as much.

The main advantage of etching the board in its current state is that we avoid the risk of scratching the toner, which is particularly high with double-sided boards. This risk can be somewhat reduced if the board is placed on a paper towel while trimming.

Furthermore, if we combined the layout for more than one circuit on the board, these individual circuits can be etched together or separately. Etching everything at once takes less time and consumes only marginally more acid, so the preferred approach is to trim the outside of the board now, and to separate ("depanelize") the individual circuits after the etching. Besides the above considerations, the best time to depanelize is also a question of the maximum board size that can be conveniently etched.

8.2.1 Cutting revisited

The cutting is done with the same instruments we used earlier for the coarse cut, see section 3.3. In order to help with the trimming, the circuit is framed by a trace that follows exactly the outline of the board. This trace can be removed when trimming, but there is no harm in leaving it intact (i.e., it will not make any unwanted connections if not removed). Note that this outline trace is interrupted where it gets too close to real traces. Do not confuse a real trace with the outline.

In order to limit the damage that can be done if accidentally scratching the board, orient it such that the least important part of the board faces away from you. In particular, if using the dremel carelessly, the mandrel may touch the board and leave deep scratches.

If using a knife to make the coarse cuts, you face the risk of the knife sliding off and cutting into the circuit. A metal ruler used to prevent this may in turn scratch the toner. Saw and dremel both are safer. If you absolutely have to use a knife for coarse cuts, consider doing the trimming after etching, when light scratches can be tolerated.

8.3 Tinning

Ideally, we would now coat the copper traces with a thin layer of tin to improve soldering and to protect against oxidation. Unfortunately, the author has so far been unable to locate the ingredients needed for this on the local market.

One approach adds tin through a chemical process. The substance used for this is called "liquid tin" and happens to be even less pleasant to handle than the acid we use as etchant. Another approach is to rub the type of solder paste used for plumb-

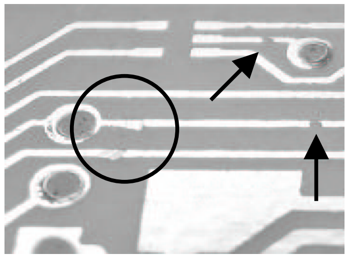

Figure 8.1: The arrows mark broken traces. The circle marks an area where the drill's mandrel has touched the board and slightly damaged the traces.

ing onto the board and to apply heat with a hot air pistol.

We discuss plastic coating as another possibility to provide some protection against corrosion in section 14.3.

8.4 Board inspection and repair

Before we start adding components, the board should be inspected for any flaws, such as broken traces or copper residues forming unwanted bridges. We can now directly measure the resistance across traces and no longer depend on guesses based on visual examination.

Most problems can also be resolved at a later stage, but at least suspiciously-looking traces and all traces that pass underneath components should be checked.

Figure 8.1 shows some defective traces: the two arrows point to fairly wide gaps in traces (gaps often are much smaller and harder to find), while the circle marks an area where the drill's mandrel has touched the board and left scratches on two traces.

8.4.1 Bridging gaps

Gaps are bridged by soldering. Soldering basics and techniques are explained in detail in the next chapter.

If a gap is very small (< 0.1 mm), you can often bridge it by simply placing the soldering iron with a bit of solder on top of the gap and "painting" solder across it.

Wider gaps usually require the addition of a piece of wire that is soldered to the trace at both ends of the gap. This wire can be isolated or unisolated. The latter is easier to handle when bridging gaps.

Unisolated wire can either be bought directly as such or it can be obtained by scavenging through-hole components, see section 10.1. Add flux to the trace, cut the wire such that it fits comfortably and does not touch any other traces or components. Then hold the wire with tweezers while painting solder along it.

Always verify through measurement that the connection has really been made. When "painting" solder, you will often be surprised to find that it hasn't.

8.4.2 Breaking connections

Unwanted connections are most easily broken with a sharp knife. Make two parallel cuts and lift or scratch off the copper between them. A single cut will often fail to reliably break the connection. Commercially available soldering tool kits often contain a steel brush to clean the trace from particles after cutting.

Always verify through measurement that the trace has really been cut. You will often be surprised to find that it hasn't.

Chapter 9

Soldering SMT components

Soldering is one of the most time-consuming parts of the production process. It is also the most interesting part, as we gradually see the circuit begin to work. This chapter briefly covers the soldering process and then explains how the individual components types are soldered.

Virtually all modern electronics use surface-mounted technology (SMT), which means that devices are soldered directly on top of the board. This differs from the traditional through-hole method, where devices have pins that are stuck through holes in the printed circuit board and soldered on the rear side.

It is a common misconception that surface-mounted devices (SMD) are, due to their smallness, impossibly difficult to solder manually. Ironically, quite the opposite is true: if employing the right methods, SMT components are easier to work with than through-hole components.

We first solder all the components on the upper side of the board, which is also called the "component" side. This way, the back side (or "solder" side) remains empty and can lay flat on the table.

All this is depicted in figure 9.1. We will work our way to the back side by adding vias and other through-hole components in the next chapter.

9.1 Safety

Solder contains lead, which is poisonous, and sometimes other hazardous substances. Therefore, wash your hands after handling solder. Hot solder can also splash. While this rarely causes problems, maintaining a safe distance and wearing glasses is still a good idea.

Flux fumes are poisonous, with nausea being a common warning sign after inhalation. Ideally, one would use a fume extractor, but working in a well ventilated area and keeping a safe distance also helps to reduce the exposure to flux fumes.

The soldering iron is very hot (typically between 270 and 300 °C, but temperatures well above 400 °C are possible), causes burns when touched, and can ignite flammable substances. In particular, keep spray cans well separated from the soldering iron. Always place the soldering iron into a stand designed for this purpose. Turn off the iron when not in use.

9.2 Soldering basics

Before we start with the actual soldering, we briefly review the fundamental concepts of soldering and the tools and materials used.

9.2.1 Soldering iron

The soldering iron melts solder and heats metal surfaces such that solder can bond them together. The heat emanates from the tip of the soldering iron and is conducted through direct contact. The contact between the tip of the soldering iron and anything it touches is greatly improved if the tip is covered by a small amount of molten solder.

Furthermore, the tip of the soldering iron must be kept free from residues that resist heat transfer, such as burnt flux and other oxidation products. Flux residues are removed by wiping the tip on a wet sponge or a ball of bronze wool. A cover of molten solder protects the tip from oxidizing. You should therefore never try to remove all the solder from the tip. Instead, add solder to keep the tip well protected.

In order to avoid damaging electronic components through excessive heat, the soldering iron should

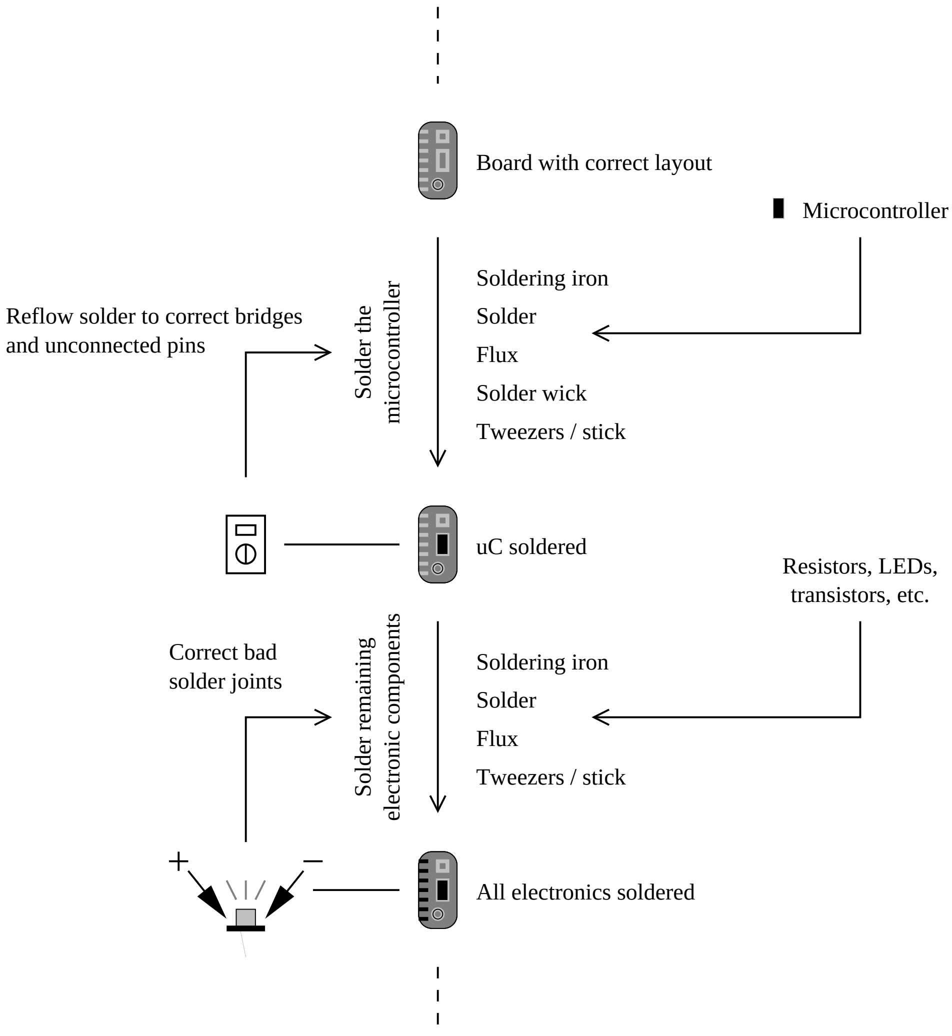

Figure 9.1: Soldering the electronic components.

Figure 9.2: Soldering irons, from back to front: professional soldering iron with soldering stand, low-cost soldering station, basic iron, and 12 V iron.

be operated at the lowest temperature that allows proper handling, which will usually be a bit less than 100 °C above the melting point of the solder. The difference determines how quickly the solder will melt when brought in contact with the tip of the soldering iron. Similarly, the time during which the soldering iron is in contact with a component should be kept to the necessary minimum.

A temperature of 270 °C is sufficient for regular components when using a lead-based solder. Lead-free solders need slightly higher temperatures. If soldering very large components, such as thick wires or sheets of metal, a higher soldering temperature may be needed. Soldering irons without temperature control typically operate at temperatures above 300 °C.

A soldering time of a few seconds it usually sufficient. If a component that is sensitive to heat, such as a semiconductor or a capacitor, needs to be worked on for a longer amount of time, you should let it cool from time to time, e.g., by alternating the pins or sides being soldered.

Excessive heat applied to copper traces can detach them from the board material. This type of accident is particularly common when trying to remove

Figure 9.3: Soldering tools. Top row, from left to right: desoldering braid, regular solder, and “eutectic” solder. Bottom row: angled and flat tweezers, flux syringe and pen.

a component that will be discarded anyway, which can yield the misconception that there is nothing left at that place that could be damaged.

Too low a temperature will cause the solder to melt only reluctantly and keep if from flowing properly. This results in fragile solder joints which may not conduct electricity at all, or fail permanently or intermittently under mechanical stress. As a rule of thumb, solder bridging two pieces of metal should always be concave in shape, and not form balls (but no rule is without exception, see section 10.1). If the latter occurs, add more heat and consider removing excess solder with desoldering wick as discussed in section 9.2.4.

9.2.2 Solder

Solder is an alloy that melts around 200 °C. When brought in contact with hot metal surfaces, it forms a mechanical and electrical bond between them. The most common type of solder is a mixture of 40% lead and 60% tin, but lead-free solders are becoming increasingly popular nowadays, mainly driven by environmental restrictions imposed in Europe.

There are several ways of applying solder: if a large quantity is needed, deposit a small amount of solder on the soldering iron, then bring the iron into contact with the two pieces of metal that should be soldered together, wait a moment to let them heat up, then add more solder while holding the soldering iron in place, until both parts are properly bonded.

It is important that both parts are properly heated throughout the soldering process, or the solder will form no or only a weak connection, which will break under mechanical stress.

Another way of applying solder is to carry all the solder needed to make the connection on the tip of the soldering iron. This is the method of choice when soldering individual pins of SMT components, where only very small amounts of solder are needed. The solder is applied by “smearing” it along the gap separating the surfaces. It will flow into this gap through capillary action. This is helped by the addition of flux, which we discuss below.

We will encounter a third method of soldering, an adaptation of automated industrial soldering processes, in section 9.3.

9.2.3 Flux

Flux is a resin that is mixed with solder. When heated, it removes oxidation from the metals, and it also reduces the surface tension of the solder. Fine strands of flux are usually included in solder wire, although one can also obtain solder that does not include any flux.

The flux included in solder is usually sufficient for relatively coarse work, such as the soldering of through-hole components, but the minute amounts of solder applied to SMT devices require the addition of extra flux. Such flux is available as a liquid and also as a gel in syringes. The latter is generally more convenient to use.

The needles to place on flux syringes are sometimes hard to find. The Chip Quick SMD removal kit, which is a somewhat expensive but quite useful tool for desoldering chips with a large number of pins, contains a small syringe with flux and the corresponding needle. The latter will also fit on larger flux syringes.

Many types of flux are corrosive and must be washed off the board after the soldering is done. Also, flux fumes are mildly poisonous and should not be inhaled.

9.2.4 Desoldering wick

Braided copper wire is used to soak up excess copper from solder joints. Traditionally, it is used to extract enough solder that the component loses mechanical adhesion and can be removed. Since SMT components will remain firmly attached with even microscopic quantities of solder, this method is less useful there. However, desoldering wick is used there to remove solder building bridges between contacts.

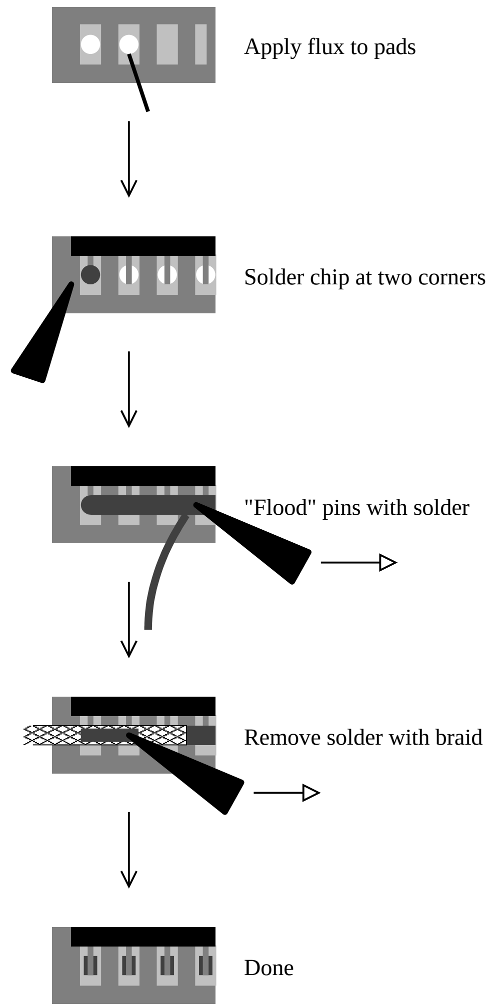

Figure 9.4: Soldering an SSOP component is surprisingly easy, given the right technique.

When using desoldering wick (also called “desoldering braid”), apply a small amount of solder to the braid to ease heat transfer. Then place the braid on the solder to remove while heating it with the soldering iron. Once the solder underneath the braid melts, it is absorbed through capillary action.

It is often advisable to raise the temperature of the soldering iron during desoldering, to ensure that enough heat reaches the solder.

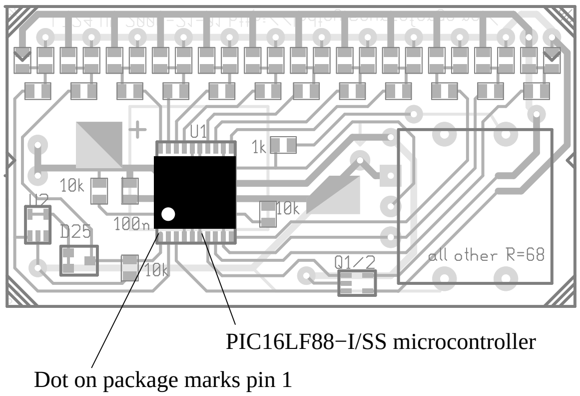

Figure 9.5: The first component we solder is the microcontroller. Pay special attention to the correct orientation.

9.3 Soldering SMT components with many leads

When contemplating the task of soldering an SSOP ("Shrink Small Outline Package") component, thoughts like "impossibly small" quickly enter one's mind. Indeed, trying to individually solder the pins[1], which are spaced at 0.65 mm intervals, is unlikely to yield anything but poor contacts, solder bridges, and a good deal of frustration.

In the most common industrial soldering processes, boards are either immersed in molten solder (wave soldering) or a paste containing solder is printed on the boards and then heated to bond with the components (reflow soldering). Although ingenious individuals [4] have found ways to bring reflow soldering within the grasp of ambitious hobbyists, neither of the industrial approaches is really suitable if we aim for simplicity and low cost.

Fortunately, there is a cheap and remarkably efficient method to accomplish that feat, as outlined in figure 9.4. The basic idea is to apply a generous amount of solder to all the pins of the chip, and then to remove the excess solder, letting capillary action retain what little is needed to connect the pins.

9.3.1 The PIC microcontroller

9.3.2 Placement

We begin by preparing the pads on which the chip will be placed, and then by attaching it to the board with two provisionary solder joints. The purpose of this is to make sure it stays at the correct place while we properly solder all the pins.

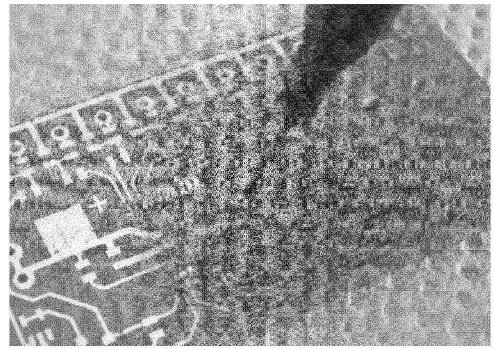

Figure 9.6: Cover the pads with flux to improve the solder's wetting action.

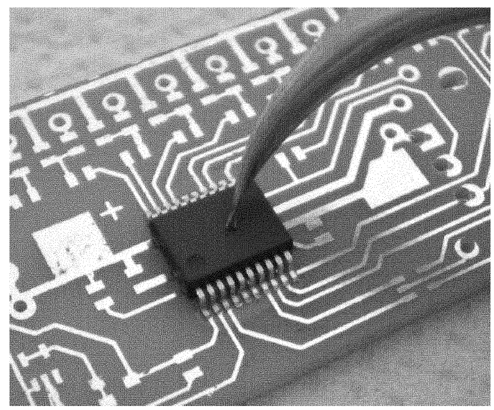

Figure 9.7: Shift the chip such that all pins are in alignment with the respective pads and traces.

Before placing the chip, apply flux to the pads. (Figure 9.6.) Flux reduces the surface tension of solder, and is necessary regardless of whether the solder already contains flux or not.

Next, place the chip on the board and position it such that all pins are perfectly aligned with the pads and traces. As shown in figure 9.7, tweezers are useful for this, as they allow gentle pushes to be administered while not obstructing the view to the pins, as fingers would.

Make sure that the chip is placed the right way around. Pin number one of a chip is usually marked with a dot, or a dent in the adjacent side of the chip

1. The technically more correct term would be "leads", but we use "pins" to avoid confusion with the metal.

not bearing pins. Similar markings can be found in the layout.

Hold the chip in place by pressing down either with tweezers or a finger, pick up a small amount of solder with the soldering iron (figure 9.8), and rub the solder against one corner pin, as shown in figure 9.9.[2] Any pin located in or near a corner can be used for this. Gently push the chip to verify that a solder joint has formed.

The chip may have shifted slightly during this. If the pin that has been soldered is still properly aligned, you can just push the chip gently back into the correct position and hold it there while soldering a pin at the opposite corner. If the chip has slided completely out of alignment, heat up the soldered pin, pull away the chip, and try again.

After soldering down a pin in the opposite corner of the first one (figure 9.10), check again that the pins are properly aligned. As can be seen in the picture, the quality of the solder joints is of little importance at this point.

9.3.3 Adding and removing solder

To solder all the other pins, cover them with a generous amount of solder by passing with the soldering iron, and melting solder from the solder wire as you go. This is shown in figure 9.11. First do one side, then turn the board around and do the other side. On the first side, start at the corner most remote from the pins that have already been soldered down, to avoid undoing the solder joint while the chip could still move.

The desired result is to have plenty of solder on and around each pin, as shown in figure 9.12.

We now remove the excess solder with the desoldering braid. Since the braid can get quite hot during this, it is recommended to cut off a small piece and to hold it with a pair of tweezers.

Hold the braid against a row of pins, then push it into the solder with the iron. (Figure 9.13.) The braid will then absorb the molten solder. Heat transfer can be improved by depositing a small amount of solder on the braid beforehand.

Once the braid has absorbed all the solder it can hold, discard it, and continue with a new piece of braid. To avoid heat damage to the chip, give it a moment to cool, and alternate sides on that occasion.

Be sure to keep on heating the braid until the moment you pull it away from the pins. If you stop earlier, the solder with solidify, bonding the braid to the board. If this happens, keep on heating until the solder is molten. Under no circumstance pull on the braid, as this can cause individual strands of copper to separate and to attach themselves to the pins, from where they can be removed only with great difficulty. (See figure **??**.)

Don't be afraid to push the braid against the pins. It will only reach all the excess solder if making good contact, as shown in figure 9.15.

9.3.4 Inspection and testing

When the solder removal is done, the result should be visually inspected. Two types of problems are common:

- Solder bridges may have formed between two or more pins, as in the example on figure 9.16. It is usually sufficient to treat the offending pins once more with desoldering braid. The braid should be half saturated with solder, to improve heat conduction. If the bridge resists, try adding more solder and/or increasing the temperature of the soldering iron. A desoldering wick of poor quality may completely fail to remove the last traces of solder between pins. In such a case, the only correct solution is to look for a better braid, as described in section **??**.

- Pins may have been left unconnected. This is particularly common for corner pins and pads from which no outward traces emanate. Such pins can sometimes be found by visual inspection. If in doubt about a pin, briefly press it with the soldering iron onto its pad, to allow the solder that will have settled on the latter to form a bond. If a this is unsuccessful, deposit a good amount of solder on that pin and clean up with the desoldering wick.

Sometimes, solder can also get splashed onto unrelated parts of the board and form connections between traces. In such cases, remove the solder with desoldering wick.

2. When taking these pictures, the author was testing the cheapest soldering iron that could be found on the market. Alas, the quality of this product greatly surpassed its price in terms of cheapness. In particular, notice that, after only a few minutes of use, the tip is almost completely covered by oxide and can hardly hold solder anymore.

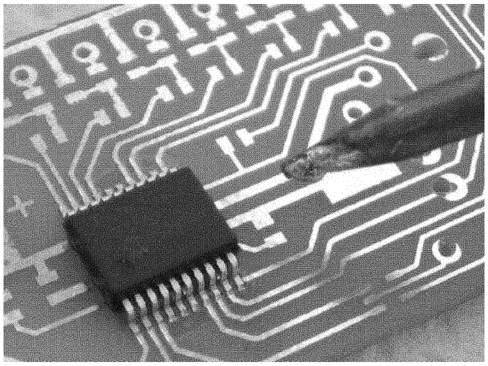

Figure 9.8: Dab a small amount of solder to the tip of the soldering iron to make the first provisional joint.

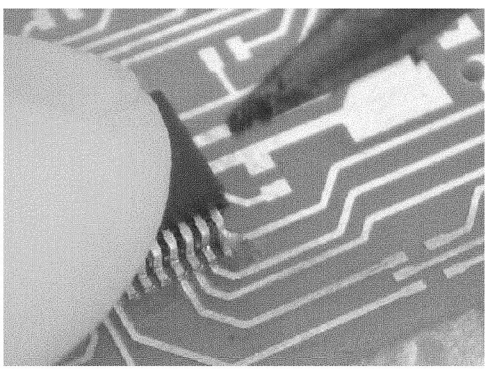

Figure 9.9: Solder a pin on one corner, ...

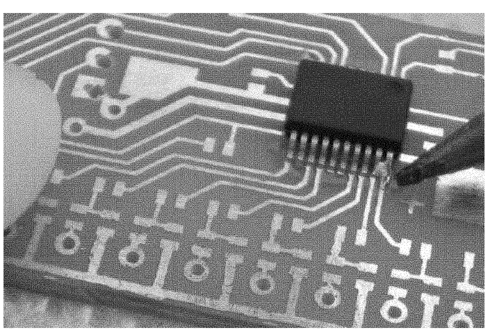

Figure 9.10: ... then fix the chip in place by soldering a pin at the opposite corner.

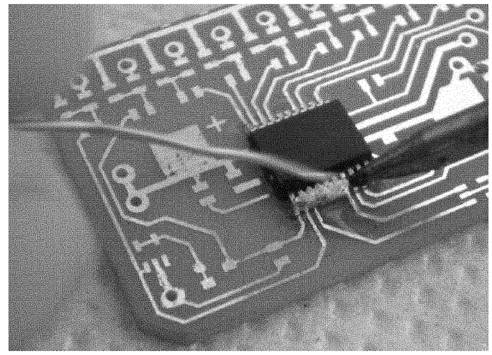

Figure 9.11: Pass along both rows of pins with the soldering iron and melt solder onto them.

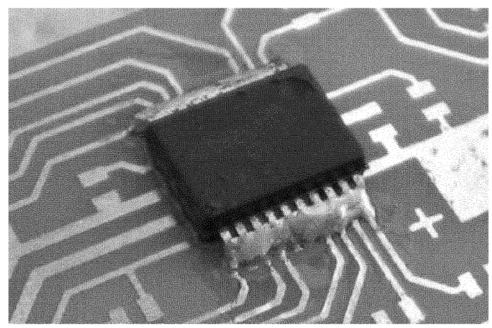

Figure 9.12: In the end, all pins should be covered thickly with solder.

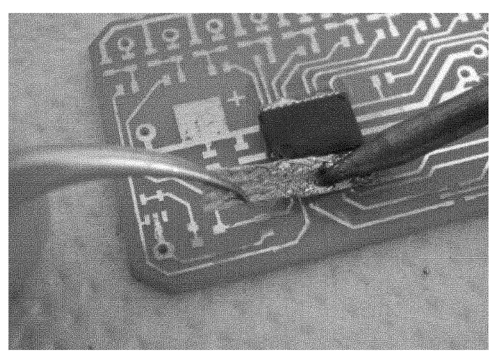

Figure 9.13: Use desoldering braid to remove all the excess solder between pins.

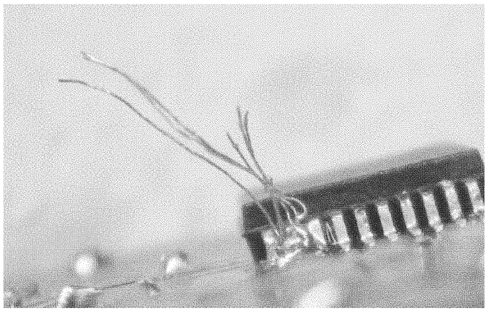

Figure 9.14: When trying to pull desoldering wick away from already cold solder, copper strands may separate, and then become very difficult to remove.

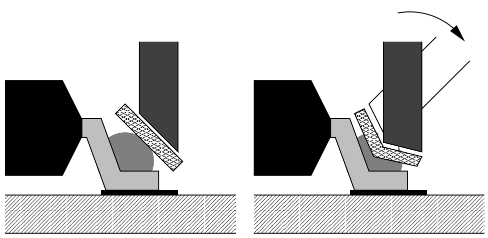

Figure 9.15: If the desoldering wick touches the pins only gently (left), it may not make enough contact to melt and remove the solder. Therefore, push against it with the soldering iron, to bend and press it against the pins (right).

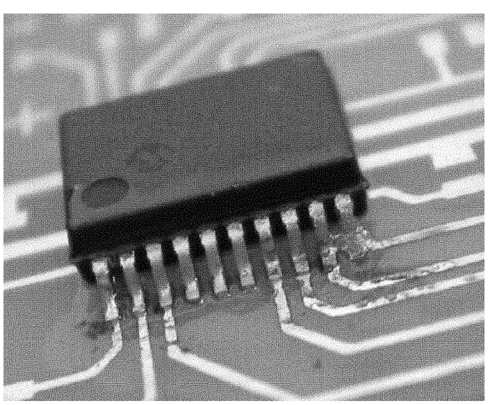

Figure 9.16: After removing the solder, solder bridges (one can be seen on the right) may sometimes remain, requiring another removal pass.

Not all flaws can be found by visual inspection. Bridges can be detected by measuring the resistance between adjacent pins or the traces they connect to.

Unconnected pins can be found by measuring the resistance between the pin and the trace it connects to. When making such a measurement, try to exert as little force as possible on the pin, to keep it from making contact only under this momentary pressure.

A quick way for checking pins is to work in one direction, e.g., from left to right, put the probe in the groove between adjacent pins and measure connectivity to the trace connected to the right pin (resistance should be low) and to the trace connected to the pin to the right of it (resistance should be high, typically in the order of > 1 MΩ, depending on connections inside the chip). Then move the probe one pin interval to the right, and repeat.

9.3.5 Component removal

If things go seriously wrong, it may be necessary to remove the chip completely and to start over again. Occasions for this are poor alignment having evaded notice until too late, the chip being soldered the wrong way around, forgotten initial programming, or some other – usually inexplicable – malfunction detected later.

To remove the chip, apply solder to all of the pins, then grab it with a pair of tweezers and lift the entire board with it (figure 9.17). This works best with crossed tweezers, where the two points push towards instead of away from each other. Now, heat up the solder such that all of it is liquid at the same time. Once this happens, the board will release the chip, and fall down (figure 9.18).

Melting all the solder at once can be tricky, particularly if there is nobody with a second soldering iron around to help. Sometimes, just rapidly alternating between both sides is sufficient. This can be helped by increasing the temperature of the soldering iron. If the solder cools too quickly, an alloy with a greater thermal capacity can be used instead of regular solder. Such an alloy is commercially available under the Chip Quick brand, see section **??**.

After removal, the board and the chip can be cleaned with desoldering wick (and a lot of patience). The chip is usually not damaged by this procedure.

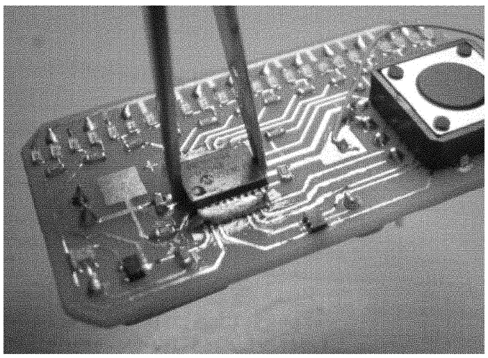

Figure 9.17: To desolder the chip, apply plenty of solder to all the pins, lift the board by pulling the chip with a pair of tweezers, and heat the solder on both sides.

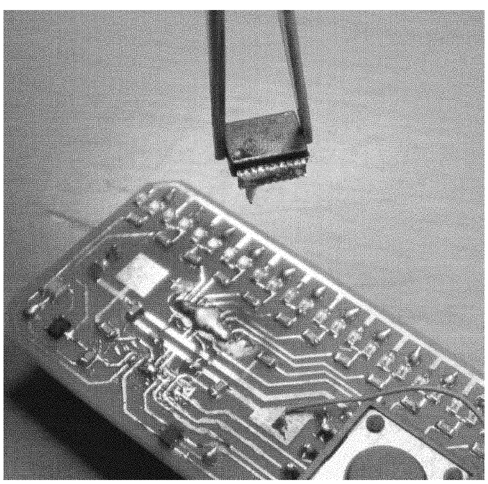

Figure 9.18: When all the solder is molten at the same time, gravity will separate the chip from the board.

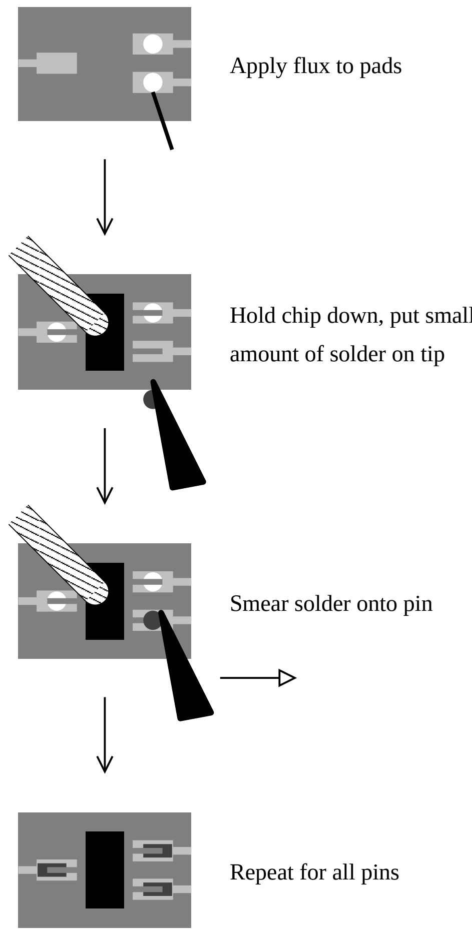

Figure 9.19: SOT components are soldered by "painting" solder along individual pins.

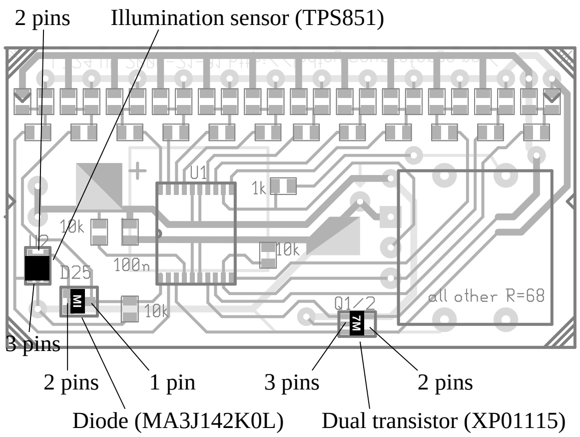

Figure 9.20: After the microcontroller, we solder all remaining semiconductors, except the LEDs. The orientation of the components can be determined by the number of pins on each side.

9.4 Soldering small SMT components with leads

After having mastered soldering the microcontroller, soldering the remaining components offers few surprises. We proceed with other SMT components that have leads (pins), such as transistors, diodes, and small integrated circuits.

These components usually come in SOT ("Small Outline Transistor") packages with three or five pins and a pin interval of 0.5, 0.65, or 1 mm.

The soldering process outlined in figure 9.19 is similar to the beginning of the process for SSOP components, described in the previous section.

9.4.1 Transistors

else: diodes, sensors, etc.

9.4.2 Component identification

Since a project may use different components in identical packages, it is a good idea to check their identity before assembly. Unfortunately, there is not enough space on the packages to print the complete product number. Instead, manufacturers print a short product code, typically only two letters or digits, which is cross-referenced in the data sheet and also in the PARTS file in the LED Toy "hardware" package.

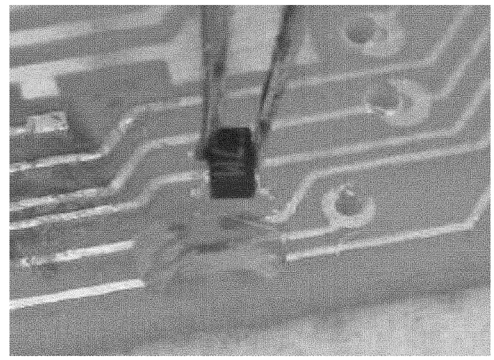

Figure 9.21: Use tweezers to drop the component onto the flux-covered pads, to shift it into the correct positions and to hold it down while soldering the first pin.

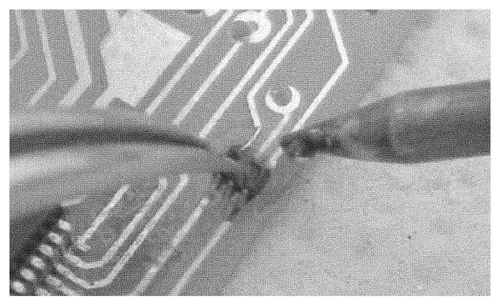

Figure 9.22: "Paint" solder onto one pin, . . .

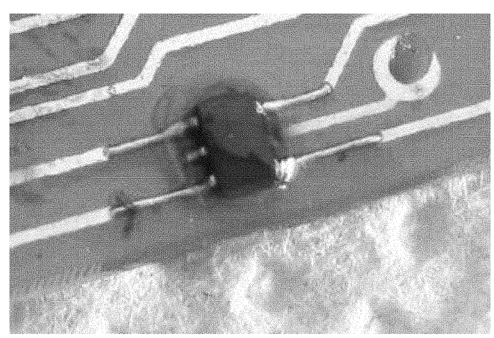

Figure 9.23: . . . then do the same with the remaining pins.

9.4.3 Placement and soldering

Before placing the component, add flux to all pads. Then pick up the component with a pair of tweezers and drop it on its place. (Figure 9.21.) Adjust the position of the component by gently pushing its sides, until all pins are on their respective pads. All this should be done with tweezers, as the component is too small for properly handling it with bare hands.

The orientation of the component can be verified by checking that each pin rests on a pad. A trace passing underneath the chip can sometimes be confused with a pad. In such cases, the layout in the PCB design program, where traces and pads are clearly distinguished, should be consulted as a reference. Figures **??** and **??** also show the orientation of all small semiconductors in the current circuit.

To fix the component in place, put a small amount of solder on the tip of the soldering iron, push down on the component to hold it in place, and smear the solder onto any pin by sliding the soldering iron over the pin, as shown in figure 9.22. Gently prod the chip to ensure that it is properly soldered down.

You can now apply solder in the same way to all other pins, without needing to hold down the component.

9.4.4 Removing excess solder

Desoldering wick can be used to soak up excess solder. Note that, if using a reasonable quantity of solder, there should be no solder that needs to be removed. Solder bridges can often be removed by simply passing with the soldering iron over both pins, which redistributes the solder.

9.4.5 Inspection and testing

The solder usually forms a long ridge that begins on the pin and extends onto the trace, as shown on figure 9.23. Its integrity can easily be verified through visual inspection.

Any remaining doubts can be cleared by measuring the resistance between the pins and the corresponding traces, like with SSOP components.

9.4.6 Component removal

A SOT component can be removed from the board by pulling on it with a pair of tweezers while heating the pins. The addition of extra solder helps to extend the time until the solder solidifies. Since the number of pins is very small, the use of special alloys is not necessary.

9.5 Soldering small lead-less SMT components

There are many types of SMT packages without leads, ranging from simple resistors to high-end processors. We focus here on simple components with two terminals, namely resistors, capacitors, and LEDs. Standard sizes include "0402" with a footprint of 0.04" $\times$ 0.02" or 1×0.5 mm^2, "0603" (1.5×0.8 mm^2), and "0805" (2×1.3 mm^2). The LED Toy uses 0603 components.

Such components are usually packaged in plastic or paper tapes, where they reside in compartments covered by a transparent plastic film. Be careful when removing the covering film: if the tape is bent and snaps back when pulling off the cover, it may catapult its content far enough that, due to its minuscule size, it will never be found again.

Figure 9.24 gives an overview of the soldering process.

9.5.1 Capacitor, resistors, and LEDs

9.5.2 Component identification and orientation

Resistors bear a number indicating their resistance. This is much more convenient than the color codes used on through-hole resistors since the dawn of time to the present day. The code consists of three decimal digits, which form a floating-point number with a two-digit mantissa and a single-digit exponent. If the digits are d_1, d_2, and d_3, the resistor's nominal value is $(10d_1 + d_2) \cdot 10^{d_3}\Omega$. E.g., a resistor with the code 102 has a nominal resistance of $(10 \cdot 1 + 0) \cdot 10^2 = 1$ kΩ.

Resistors can be soldered in either orientation, but it generally looks better and helps visual inspection if resistors are soldered such that the text on all resistors with parallel main axes face in the same direction. Non-electrolytic capacitors are not labeled and can be soldered in either direction.

Electrolytic capacitors (which are presently not used in the LED Toy) and LEDs have a positive and a negative terminal and must be soldered with the correct orientation. LEDs have a visible orientation

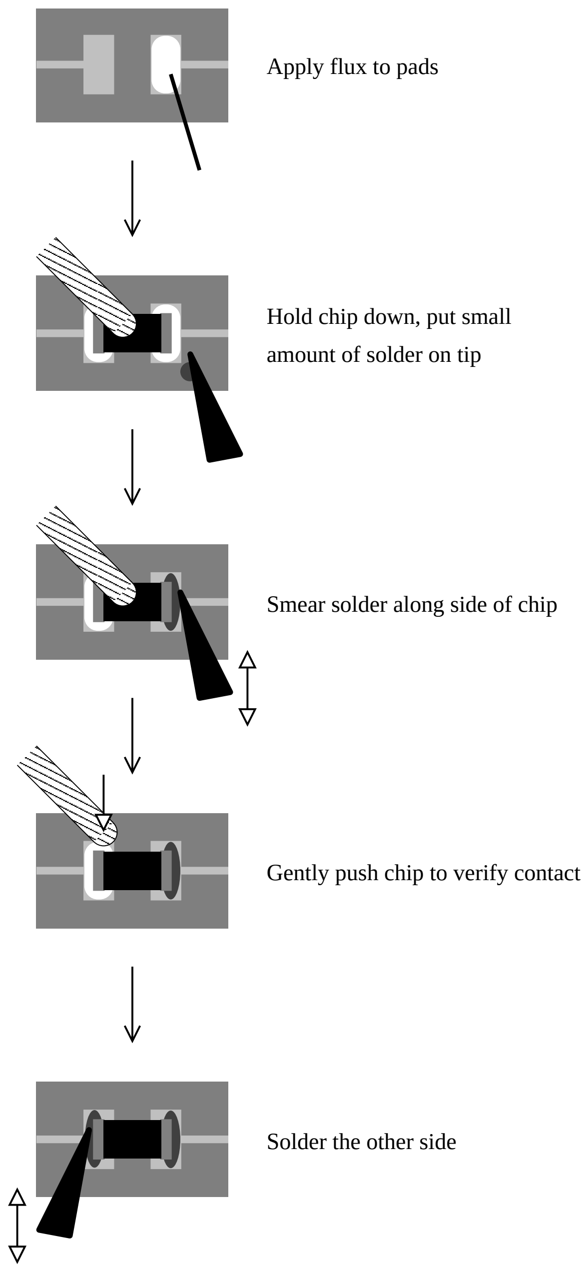

Figure 9.24: Components without pins (e.g., in 0603 or 0805 packages) are soldered by “painting” solder along the connecting surfaces.

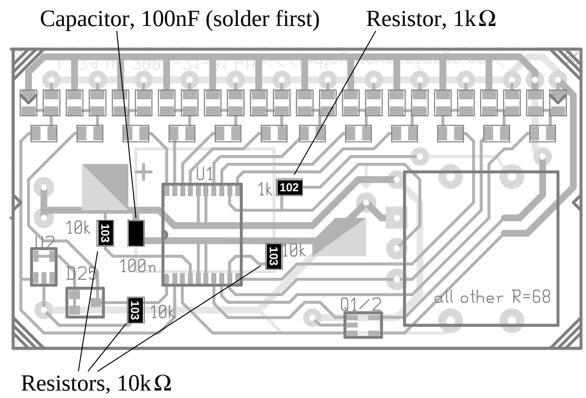

Figure 9.25: ...

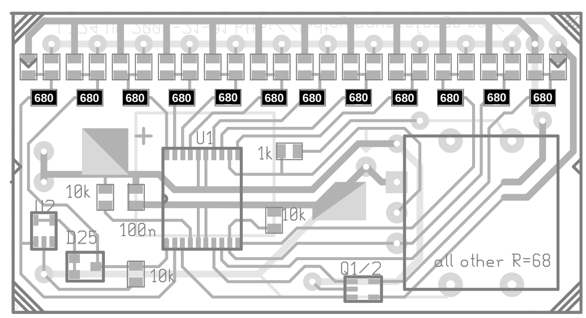

Figure 9.26: ...

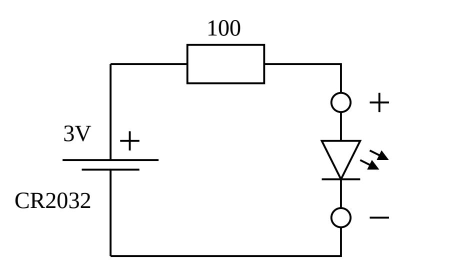

Figure 9.27: Simple test circuit to test the polarity of a LED. If using a different battery voltage, adjust the resistor accordingly. Do not use a battery that produces more than 4.5 V.

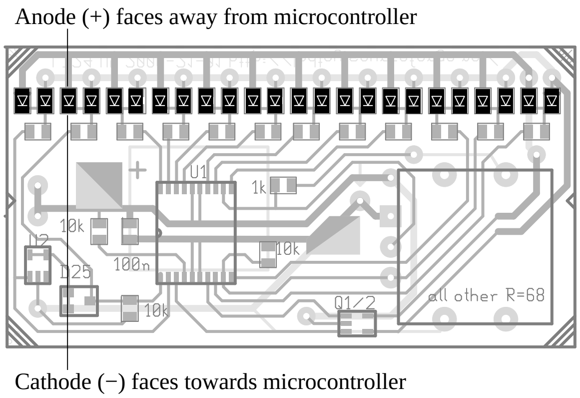

Figure 9.28: ...

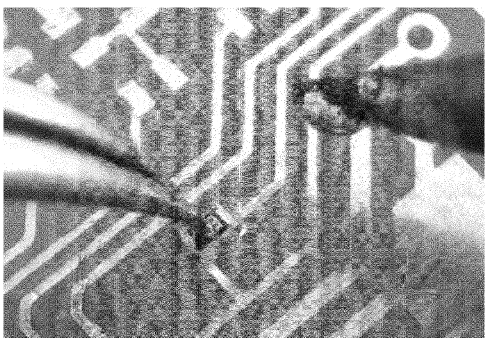

Figure 9.29: Use tweezers to drop the component onto the flux-covered pads, to shift it into the correct positions, and to hold it down while soldering one side.

but often bear no obvious marking that indicates their polarity.[3] The easiest way to determine the polarity of a LED is to apply a voltage across its terminals and to see when it lights up. Figure 9.27 shows a simple test circuit. The 100 Ω resistor limits the current to roughly 10 mA.

In the LED Toy circuit, the positive terminals of the LEDs are next to the board edge, facing away from the microcontroller. This is shown in figure 9.28.

9.5.3 Placement and soldering

Apply flux to the pads, pick up the component with tweezers, drop it on the pad and push it into the correct position, then hold it down and paint solder on one side, as shown in figure 9.29. Since this type

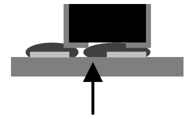

Figure 9.30: A solder bridge may form between both terminals under a component placed off-center.

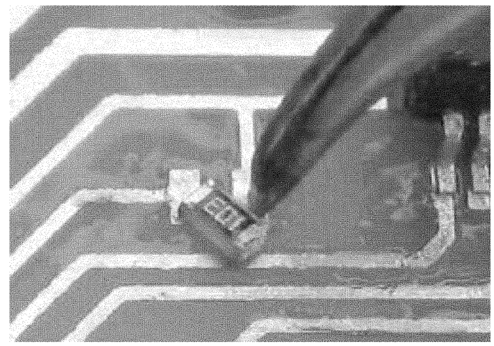

Figure 9.31: Gently push against the component after soldering, to verify that a solder joint has formed. In this example, no solder joint has formed.

of component almost completely fills its pads, you may have to try a few times until a visible bridge has formed.

If a component shifts from its position during soldering or if an adjacent component gets relocated, don't try *ad hoc* adjustments. Instead, put away the soldering iron and return the components to their rightful places with tweezers.

During all this, it is important that the component is properly centered between the two pads. Otherwise, an invisible solder bridge may form under it, short-circuiting both terminals, as shown in figure 9.30.

After that, gently push against the component (figure 9.31) to verify that the solder has indeed formed a solid joint. Then proceed with the other side. Note that some components conduct heat readily to their other end, so working too long on the second side may also melt the solder on the first one.

If the pads fit the component very tightly, it can be difficult to make a proper solder joint between

3. For example, a colored line on the package may indicate the positive *or* the negative terminal, depending on the product.

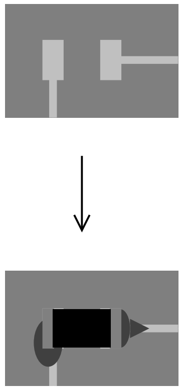

Figure 9.32: If the pads are very small, some improvising may be needed.

pad and component. Moreover, it is hard to tell through visual inspection whether capillary forces have helped to make the joint or not. In such cases, it can be necessary to use adjacent traces as additional contact surface, as shown in figure 9.32.

In general, if the placement of a component suggests that soldering will be difficult, start with the more difficult or less accessible side first. If soldering fails, you will detect this immediately afterwards, when pushing against the component, and don't have to wait until functional testing.

It is usually best to solder components in groups, e.g., start with all resistors that are not part of a regular structure, then all capacitors, followed by the remaining resistors (i.e., the ones next to the LEDs), and finally the LEDs themselves. In each wave, first apply all flux, then solder all first sides, etc. That way, it is easy to properly align components with respect to each other, which is particularly important for the LEDs.

9.5.4 Component removal

Removing lead-less components can be tricky, because it is difficulty to rapidly apply heat to both sides while holding them firmly with a pair of tweezers. A special device called "hot tweezers" is used for professional repair work. This is basically a soldering iron with two tips, which grabs and heats the component on both sides.

Lacking such sophisticated machinery, the easiest way to separate a lead-less components from its pads is often to put a bit of solder on the iron and to push against the *side* of the component. The component will heat up as a whole, the solder on the pads will eventually melt, and the pressure of the iron will push the component away.

If the component ends up on a trace or another pad, it may bond to it, and the procedure may have to be repeated. If the component attaches itself to the soldering iron, it can be picked off with tweezers, or wiped onto the sponge or bronze wool.

Since the component will be evenly heated to well above 250 °C, it may be damaged or destroyed by this removal method.

Chapter 10

Through-hole components

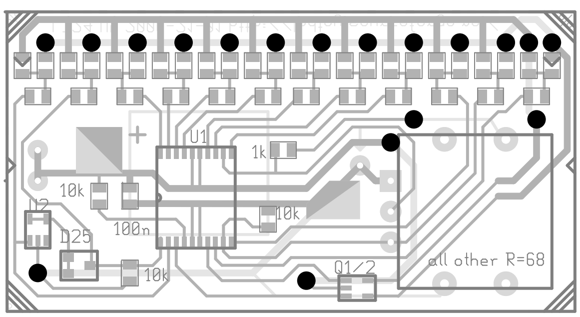

Figure 10.2: ...

After that, all connections on the board are present and can be tested.

The final section of this chapter covers the few remaining components and cleanup work.

10.1 Making vias

10.2 Electromechanical components

Only a few electromechanical components remain to be soldered.

10.2.1 Electromechanical components

10.3 Finishing touches

10.3.1 Flux removal

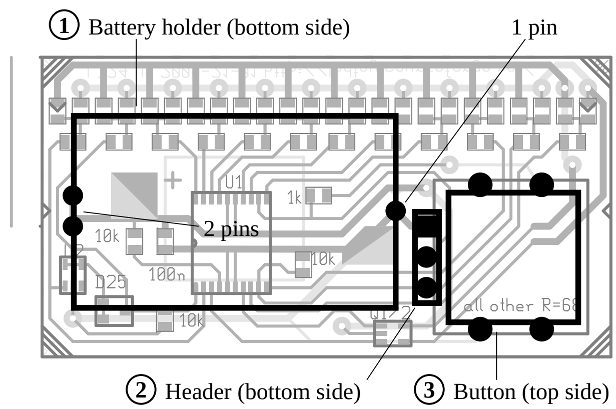

Figure 10.3: ...

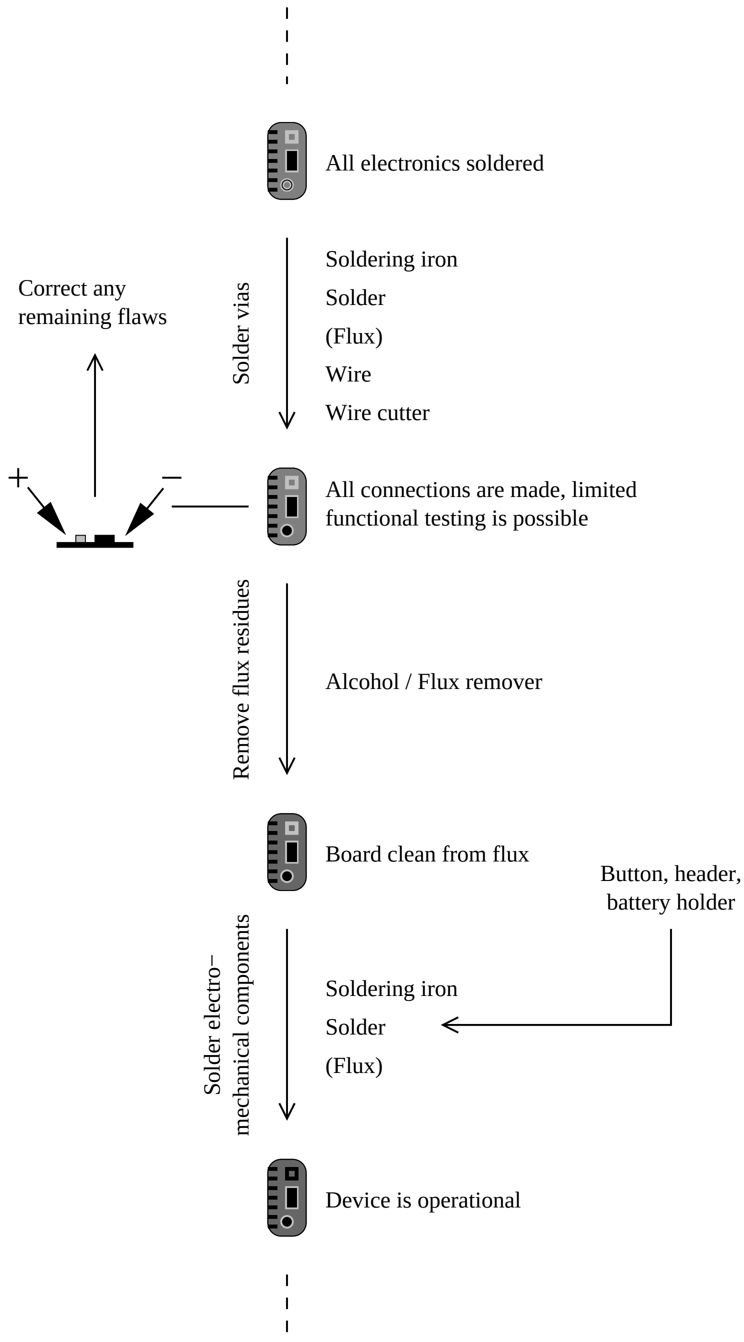

Figure 10.1: Soldering the electro-mechanical components.

Chapter 11

Serial cable

11.1

11.2 Programming

Chapter 12

Testing

When trying to power on the board for the first time, the most common result is that something doesn't work. Worse yet, defects can also develop after initial testing. This chapter describes thing that can go wrong, how to find them, and how to repair them.

This complements and often duplicates the tests that should be done after the various production stages. In general, if a problem can be identified early on, it will be easy to determine its cause, and to repair it. The longer repair or even diagnosis is delayed, the more other elements will get involved, which will complicate analysis (since there are now many more possible causes of the problem), and may get in the way of repair (e.g., because the place that needs repairing is now difficult to access).

Before you start testing, make sure to have a print-out of the assembly (see section 4.4) and a permanent marker at hand. Use the permanent marker to indicate where the problem spots are on the board and write down further details on the assembly sheet.

12.1 Types of problems

12.1.1 Interrupted traces

12.1.2 Shorted traces

12.1.3 Bad solder joints

12.1.4 Shorted components

12.1.5 Reversed LEDs

12.2 Resolving problems

12.3 Failure patterns

When powered on, the circuit rapidly runs through all LEDs, from LED 0 (at the top) to LED 23 (at the bottom). This gives a first idea of how well the circuit works. The LEDs can be activated more slowly by pressing the button, stepping through the menu (see [1]), selecting the "Timeout" function, and then rotating through the LEDs with individual button presses. In this mode, the LEDs are activated from bottom to top, and loop back from top to bottom.

In the following sections, we discuss various failure patterns and their possible causes. In each case, we assume that a single defect is the cause. The purpose of all this is to convey a sense of where to look for common defects and their often surprising symptoms.

Particularly if building a board for the first time, there can be a fairly large number of similar defects, which complicate analysis. In this case, you should test all the components first, as described in section 12.1, repair all problems you find, and only then look for patterns.

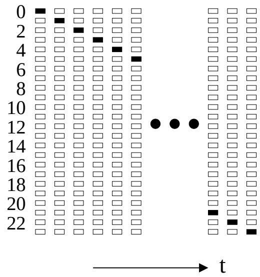

Figure 12.1: ...

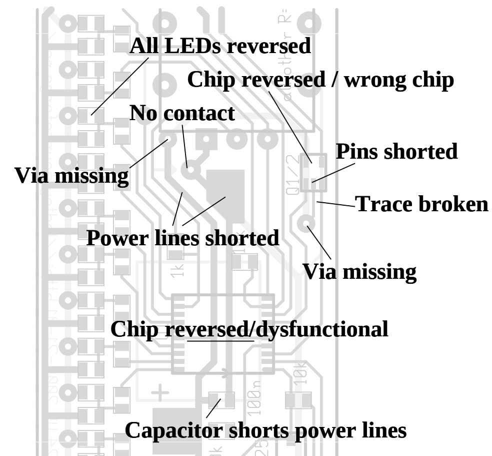

Figure 12.2: ...

12.3.1 Applying power

The first step is to see what the circuit does when it receives power. This can either be done by inserting the battery (if the battery holder is already in place), or by connecting a power source to the two large power supply pads, as shown in figure **??**. The power source should supply 3 V and 100 mA. If the power source has a variable current limit, set it to 100–200 mA.

The LED Toy should now display the walking LEDs and then power itself down. We use a diagram like the one in figure 12.1 to indicate the pattern shown on the LEDs: the line of LEDs is shown vertically, and the changes over time are shown horizontally. If stepping through LEDs manually in "Timeout" configuration mode, the direction is reversed, i.e., the matrix with the patterns has to be read from right to left.

12.3.2 Strange, nothing happens

Quite often, the circuit just stays dark. This can mean that the microcontroller does not receive power, that it does not start, that it has no program to execute, that power does not reach the LEDs, or that the LEDs can't use it.

Figure 12.2 shows a selection of possible defects that would cause the circuit to remain dark. There may also be multiple defects, e.g., if both traces from the transistor array to the LEDs are broken,

Reversed components

The first thing to check is that all the components have the correct orientation, see figures 9.5, 9.20, and **??**. Of all mistakes you can make, this is one of the most unpleasant ones, and it is good to get this worry off your mind early on.

If you have put all LEDs the wrong way around, you can resolve this in one of two ways: (1) you can remove all LEDs and solder them back on in the right way. This is quite a lot of painful work. (2) if you feel confident, you can convert the circuit such that current flows through the LEDs in the opposite direction, as follows:

- Replace the XP01115 (PNP/PNP) transistor array with an XP01215 (NPN/NPN) array
- Cut the trace to the common emitter pin of the transistor array (that's the middle pin on the side with three pins). This is best done on the bottom side of the board, so that space is left for re-wiring.
- Wire the common emitter (see figure 12.3, which shows the PNP counterpart, but with the same internal structure) to ground
- Modify your firmware to set `HW_SINK`[1]

Before starting to turn around components, triple-check that they really have the wrong orientation. This is a fairly intrusive type of repair with a high risk of damaging components and causing other problems, so it should only be attempted if absolutely necessary.

1. Remember to make this change also when updating the firmware later. Without the change, the image will appear inverted. Also note that the LED pattern on the LEDs will look strange when using the serial port.

If you happen to have other chips with the same type of package at your work place, also consider that you may have picked the wrong one. This is particularly easy for the transistor array. You can check for this by examining the code printed on the package with the number shown in figure 9.20.

No power to the circuit

Power can fail in two ways: there can be a short-circuit, or power can be absent in the first place. Short-circuits can be found by measuring the resistance between the positive and negative supply lines. If the resistance is very low (typically $< 1\ \Omega$, but anything below $100\ \Omega$ is suspicious), we have a short-circuit right across the power supply. See sections 12.1.2 and 12.1.4 for how to find short-circuits. One not very obvious location of a short-circuit may be across the capacitor, see section 12.1.4.

Another reason for a lack of power can be that the connection to the power source is interrupted somewhere. This could be a broken trace, an improperly inserted battery, or a poorly soldered battery holder. If power can be supplied during testing (i.e., if the battery holder is in place or if you have soldered wires to the power supply pads to permanently attach an external power supply while keeping your hands free for doing the measurements), check the voltage across the points marked in figure **??**. If there is no power during testing, check the resistance between the points shown in figure **??**.

In case you have an oscilloscope: if the microcontroller receives power and is operational, you should be able to see activity on pins **RA4** and **RB0** after power-up. If the microcontroller tries to communicate over the serial port (see section **??**), there should be a square wave with a period of 4 seconds and a duty cycle of 75%. If the microcontroller starts regular operation, you should see 12 periods of a 10 Hz square wave with a duty cycle of 50%.

No power to the LEDs

The circuit also stays dark if only the power supply to the LEDs is broken. A single point of failure for this is path from the positive pole of the battery to the common emitter (see figure 12.3) at the transistor array. Besides the traces themselves, weak points include the vias in that path, as shown in figure 12.2.

If the bases of both transistors are shorted to the emitter, the transistors remain "off". You can test for a short-circuit between emitter and bases by measuring the resistance between the traces connecting to them.

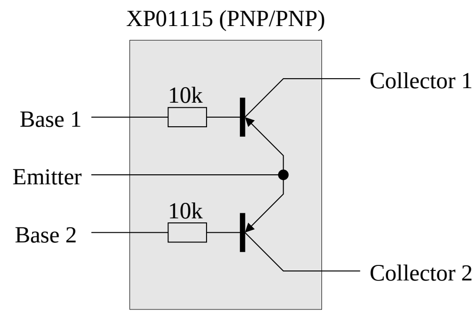

Figure 12.3: The XP01115 transistor array contains two PNP transistors, with a 10 kΩ resistor on each base, and with a common emitter.

12.3.3 The first two LEDs blink or stay lit

If the first two LEDs are lit and blink briefly every four seconds (figure 12.4), the microcontroller "thinks" it has detected that the serial link is connected.

This can happen if the serial input is shorted to a line with high potential, or if the serial input pin of the microcontroller is not pulled down to ground through the 10 kΩ resistor.

The former can be found by the usual testing for short-circuits. The latter is found by testing the resistance between the **RB2** pin and ground. It should be between 9 and 11 kΩ.

If the two LEDs stay on all the time, the microcontroller probably starts, (erroneously) detects the serial interface, and then resets itself and starts over again. This is one possible result of having a short-circuits between some of its pins.

12.3.4 A single LED stays dark

A fairly common problem is that one or more single LEDs stay dark, as shown in figure 12.5.

The most common cause for this is a bad solder joint at one of the LED terminals, which can be detected using one of the procedures described in section 12.1.3. Another possibility is that the LED is reversed (section 12.1.5) or that its terminals are shorted (section 12.1.4).

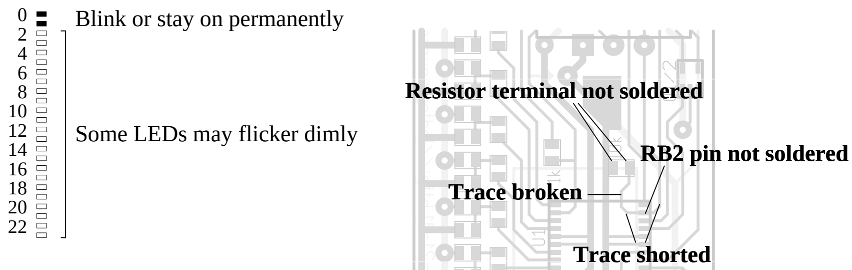

Figure 12.4: ...

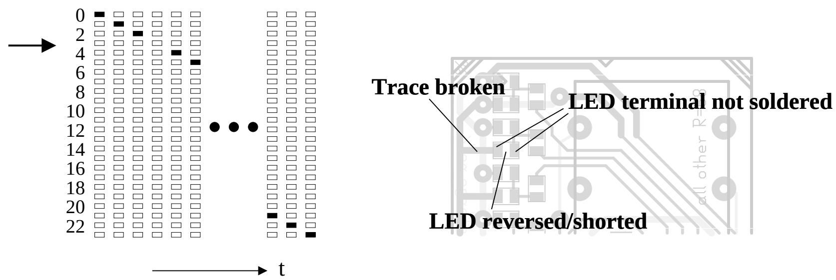

Figure 12.5: ...

A different type of cause is a broken trace, typically at the anode of the LED (see figure 9.28). Similarly, if this is an even-numbered LED, its anode connects to a via, and this via may have a defective solder joint. All this can be probed for by applying power across the LED, as described in the sections mentioned above.

12.3.5 A pair of LEDs stays dark

Another common problem is that a pair of LEDs stays dark, as shown in figure 12.6.

The reason for this is usually a badly soldered resistor, i.e., with one of its terminals unconnected (section 12.1.3), or the terminals shorted (section 12.1.4). Another possible cause is that the LED pair is not connected to the corresponding I/O line of the microcontroller, e.g., because the trace is broken or because the pin of the microcontroller is not properly soldered.

In the case of the LEDs 2 and 3, the trace passes through the header for the serial interface. Mechanical stress applied to the connector can cause the trace to break after some time of use. This can be repaired by either adding solder, to bridge the gap, or by running a wire around the button, as shown in figure **??**.

12.3.6 LEDs light up in pairs

If LEDs light up only in pairs, as shown in figure 12.7, there is a short-circuit between the two lines selecting the phases.

Such a short-circuit can easily appear at one of the numerous vias between the two lines, particularly in the upper edge where three vias are closely spaced together. Less common causes would be a short-circuit between the traces connecting to either side of the transistor array, or a solder bridge across its collectors.

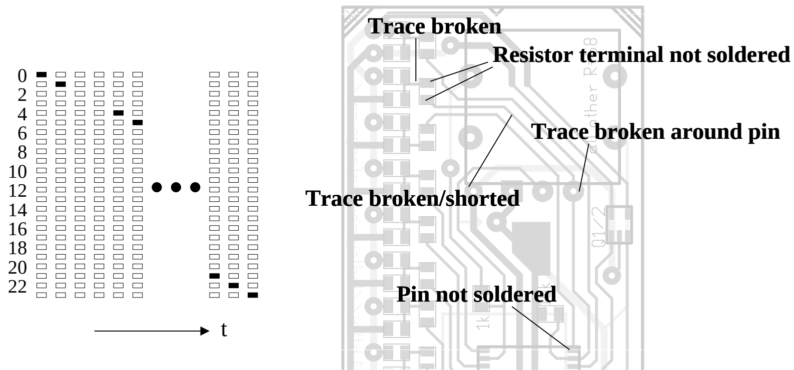

Figure 12.6: ...

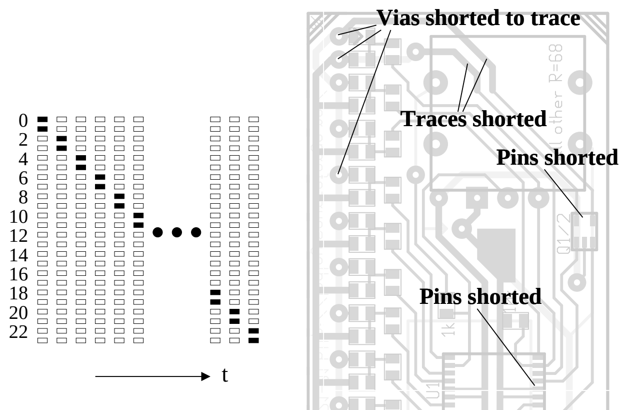

Figure 12.7: ...

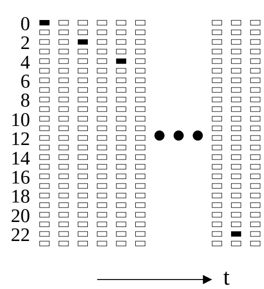

Figure 12.8: ...

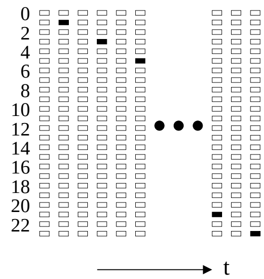

Figure 12.9: ...

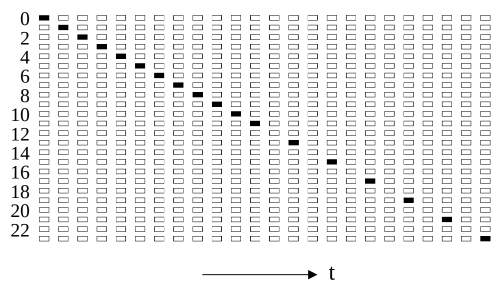

Figure 12.10: ...

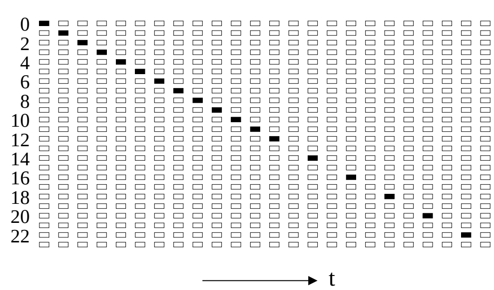

Figure 12.11: ...

12.3.7 Half of the LEDs stay dark

12.4 Board testing

12.4.1 LEDs

12.4.2 Resistors

12.4.3 Traces

12.4.4 Other small components

12.4.5 The microcontroller

- fix all visible problems - correct "odd but apparently good" components (e.g., 0603 turns easily) - short-circuits (e.g. joints, vias) - short-circuit under component - open connections, bad joints - if stuck in reset, short pull-down - if no button, short contact - lead-covered holes - re-check connections - check LEDs with electricity - if external power available, use it (avoid adding battery holder) - add header last, button late

- use assembly sheet to mark flaws

- buying extra components: (+10- drilling: walls help

- remove bridges from shorted parallel traces with flux - bridge interrupted traces with solder (no flux !)

goals: a) help people with prior knowledge to master modern techniques b) give new starters an idea (probably not sufficient for self-study)

Chapter 13

Building the case

We come now what may be the most grueling task of whole project: building a nice-looking case for the LED Toy. The type and shape of the case depends on personal taste and also on the material available. This chapter describes how to fit the LED Toy into the "GABMA" plastic case available at Electrocomponentes S.A. (see appendix A for details), which is a simple and inexpensive box of roughly $52 \times 30 \times 16$ mm^3.

If you want to use a very different case, you may have to adjust the size of the circuit board. It is also possible to choose different case designs with the same form factor. This is discussed in section 13.2.

Figure 13.1 shows the parts that go into the case. From bottom to top: the case with a hole cut into its bottom, a transparent front cover, a sheet of paper with the LED numbers and menu positions, the circuit, a CR2032 battery, and the back cover. All this, except the back cover, is held in place by a metal clip, which also serves as means for attaching the LED Toy to clothes . The circuit and the clip are separated by the two pieces of paper shown on the right.

13.1 Safety

Besides the by now familar cutting and cleaning tools, we now encounter two new items: coiled wire and instant or "super" glue.

13.1.1 Wire

For the clip, we use relatively thick wire. This presents two hazards: coiled wire may spring back when cut or otherwise released, and cut pieces of wire may fly away in an arbitrary direction. The sharp endings of the wire are thus a danger to eyes.

13.1.2 Instant glue

Instant glue is normally based on cyanoacrylate. Cyanoacrylate is toxic and gives off fumes that irritate the eyes and may cause breathing difficulties. Avoid contact of the glue with skin and eyes.

Note that the fumes reportedly get particularly intense and dangerous if the glue is heated.

- cut rectangular hole in bottom of box - extend hole so that all LEDs are visible - cut cover from soft transparent plastic - trim cover - hold over toy, mark location of button - cut hole for button - extend hole for button - print cover paper - cut cover paper - optional: glue down cover paper - make clip holes into box - clean wire - bend wire - clean wire and box - cut isolation papers - assemble - fix with super-glue - let dry for at least 2 hours - cut extra wire and file off sharp points

13.2 Case evolution

The design presented in this chaper is not the first one that was tried. This section reviews some of the alternatives, and explains why they were ultimately rejected.

An earlier version of the case design, shown in figure put the circuit such that it faced the top of the case, not the bottom. In that design, the original cover of the case was replaced with one made of acrylic glass ("Plexiglas"). The internal wires of the clip ran next to the bottom of the case, where there were held by epoxy glue. There was an optional hole for the serial connector. In order to keep the circuit board from tilting when the button was pressed, a

piece of plastic was glued into the case underneath the button.

This design looked quite nice, with the transparent surface covering the whole front. Since the case is larger at the top than at the bottom, a LED Toy placed on a table would “look up”.

However, there were several weaknesses. The worst problem was that the acrylic glass was what held everything together, so it had to be cut with extreme precision, which is made worse by acrylic glass being hard to work with and very brittle. Unfortunately, even a perfectly shaped front plate would fall off when the LED Toy was dropped to the floor, spilling the contents (cover, paper, circuit, and the battery would usually separate, too), and sometimes the cover would break on that occasion.

Replacing the battery was inconvenient, because the whole assembly had to be taken apart to reach it.

The current design improves upon this in many ways. Since the front is now held by the remains of the bottom of the original case, the front plate does not have to be very precise, and can be made of a soft material. The back of the board is now held safely by the clip. The clip is attached to the case at only four points, which simplifies application of the glue. When the LED Toy falls to the ground, only the rear cover and the battery can separate easily, but even this is less likely, because there is no longer the weight of the whole circuit pushing against the cover. Even if some of these parts are lost, they are easily replaced.

However, also the current design has its weaknesses. The removal of most of the bottom of the case makes it less rigid, which can be easily observed when attaching the clip. Since there is now a frame of case material before the transparent front plate, the first and the last LED are sometimes difficult to see. Last but not least, it is virtually impossible to extract the circuit for repair, without destroying the case.

Possible future improvements include replacing the button with a capacitive sensor, which would greatly simplify the making of the front cover, and it would also eliminate an often unreliable electromechanical component. Another change worth considering would be to cut a thread into the endings of the clip, and fix it with nuts on the outside of the case, instead of using glue. This way, the clip could be removed if repair is needed.

Chapter 14

Process improvements

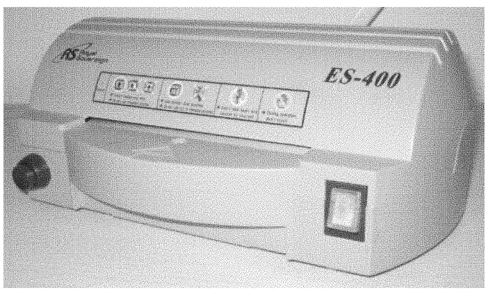

Figure 14.1: A cheap laminator can replace the clothes iron in the toner transfer process.

The focus of this document so far has been on methods that require only tools and materials that should be easily available to anyone interesting in building electronic circuits.

There are a few slightly more advanced tools and concepts whose application can yield dramatic improvements in quality, efficiency, and convenience, while still not costing a fortune. In this section, we discuss some of these improvements.

We have already encountered one such advanced tool, namely the dremel. Since it makes all the cutting, drilling, and smoothing so much easier compared to the use of exclusively manual tools, and it has many uses beyond electronics, it is recommended as a standard tool, even if it is not extremely cheap. The following tools are more specialized.

14.1 The toner transfer laminator

Toner transfer uses heat and pressure to copy layouts from sheets of plastic-coated paper to the copper of a PCB. One problem with this transfer process is that the results of manual ironing can vary quite a bit. Therefore, it would be desirable to have a means to automate this, such that more consistent quality can be achieved.

Fortunately, there is a type of machine that combines heat and pressure in much the same way we need in this case: the laminators used to glue plastic sheets coated with a heat-activated adhesive to paper or similar material. Small laminators, like the one in figure 14.1 are relatively inexpensive and can be found at shops selling office machines.

Such a laminator will usually require some modifications before it can be used for toner transfer. Recommendations for specific models can be found on the Web and some companies also re-sell laminators where the modifications are particularly easy. Here, we describe the general concepts behind all this, allowing the reader to choose a suitable device and the exact set of modifications.

Warning ! Some of the modifications will probably require you to open the laminator and perhaps even to change its wiring. Parts inside the laminator are at mains voltage, i.e., up to 240 V AC. Exercise due caution. In particular, disconnect the laminator from power before working on it.

14.1.1 Principle of operation

Figure 14.2 illustrates how a laminator works: the paper and the two plastic sheets are pulled through the laminator by a pair of motor-driven transport rolls. These transport rolls apply pressure, and are heated by two heating elements, one above and one below the paper, so they also transfer heat to the plastic sheets. The heat melts the adhesive and fuses the plastic sheets to the paper.

The temperature of the heating element is con-

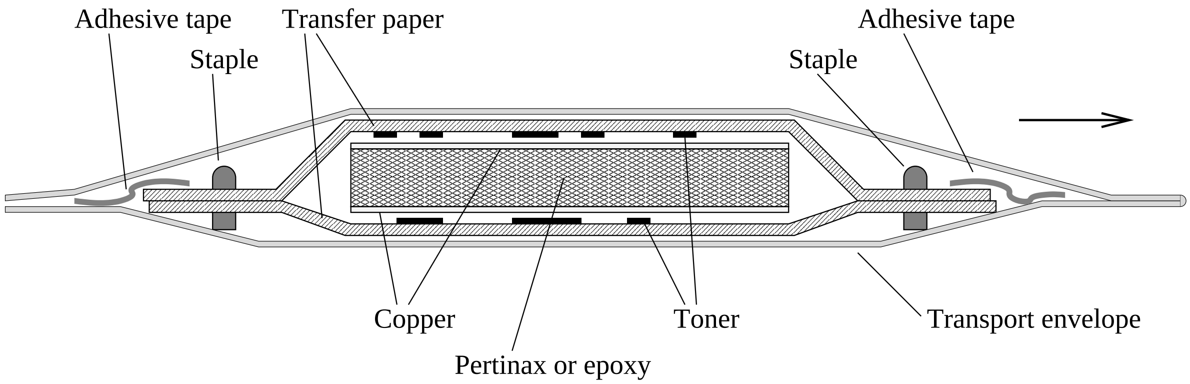

Figure 14.3: The paper stack used for toner transfer with a laminator. (Figure not to scale.)

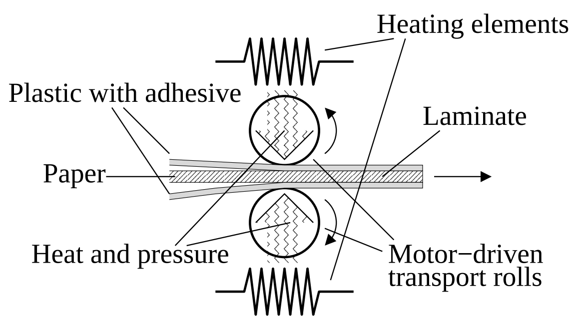

Figure 14.2: Principle of a laminator: heat and pressure are applied through the transport rolls, which melts the adhesive on the plastic and fuses it to the paper.

trolled by a thermostat. In simple laminators, this temperature and the speed at which the paper is transported are usually fixed.

14.1.2 Paper stack and thickness

Figure 14.3 shows the paper stack we use with a laminator. It is very similar to the one used for manual ironing, with two exceptions: (1) instead of just wrapping the transfer package in a sheet of paper, we must now carefully fold this paper and cut it to size, such that it can be properly transported through the laminator. We call this paper the transport envelope. (2) since pressure is not applied straight from the top, we tape the transfer package to the transport envelope, to prevent it from shifting.

The transport envelope not only serves as a means for pulling the whole stack through the laminator, but it also protects the laminator from any sticky materials that may come loose from the transfer paper.

Unfortunately, this whole package can get rather thick, up to maybe 3 mm. Most laminators are not designed to handle paper this thick. This can cause three types of problems: (1) the paper path may be too narrow, (2) the transport mechanism may not be able to pull in the board, and (3) the laminator may get damaged by all this.

Thankfully, (3) does not seem to happen very often. (1) usually takes the form of a ruler, i.e., a flat piece of metal meant to keep the paper from curling and going astray, blocking the exit. You will notice all such obstructions when passing a trial stack through the machine. Guides can often be easily removed after opening the laminator.

(2) is a bit more tricky. First, in order to make the transport mechanism "take" the board, the stack may have to be pushed or pulled manually. Pulling is preferable. To prepare for this, use an extra-long strip of paper for the transfer envelope, such that a few centimeters of the envelope will have left the laminator by the time the board reaches the transport rolls. When the stack gets stuck at the edge of the board, you can simply pull a little to help the motor. After that, the transport mechanism can proceed on its own.

The second problem is that the heat transferred to the board is not properly controlled if the board is manually pulled between the rolls: first of all, while the stack is stuck, more heat than usual may be transferred to the edge of the board. Then, when

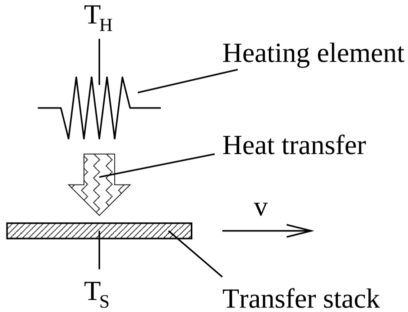

Figure 14.4: The temperature in the transfer stack depends on the temperature of the heating elements and on the speed at which the stack is transported.

the stack is being pulled, it moves quickly for several millimeters, thus reducing the heat. It is therefore best to allow for temperature to stabilize by leaving a generous margin of about 1 cm at the beginning of the board, before the actual circuit begins.

14.1.3 The right temperature

Toner transfer normally needs more heat than a laminator delivers to the stack. To solve this problem, we have to consider the following parameters, shown in figure 14.4:

T_T is the temperature at which toner transfer occurs with optimal results

T_S is the (peak) temperature inside the stack

T_H is the temperature of the heating elements

v is the speed at which the stack is pulled past the heat source

Our goal is obviously to achieve $T_S \approx T_T$. With an unmodified laminator, T_S tends to be relatively low. So we can try to choose materials where only a low T_T is needed, search for a laminator that allows us to set a high T_S, or try to increase T_S in a laminator we have already chosen. The rest of this section describes the last approach.

There are two ways to increase T_S: (1) increase T_H, or (2) reduce v. Increasing T_H can be dangerous, because it will cause the laminator to operate outside its designed temperature range, which can cause it to malfunction, shorten its lifespan, and even cause a fire.

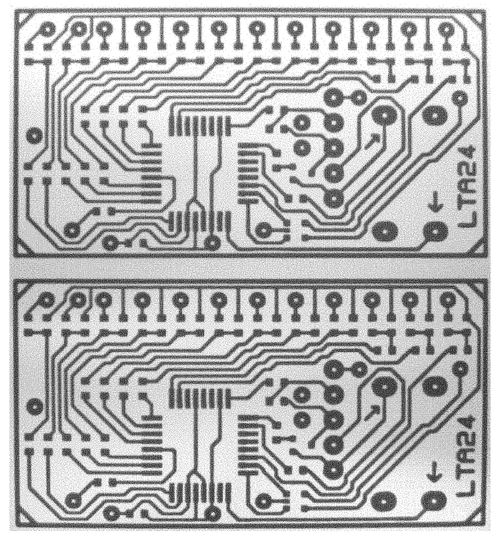

Figure 14.5: As this example shows, a laminator can help to produce results of a quality virtually unattainable with manual ironing.

Decreasing v is a much safer approach. For this to be effective, T_H must be greater than T_T. If the board moves more slowly past the heat source, more heat will be transferred, and the temperature will eventually reach the desired level.

One way to reduce v is to periodically cut power to the motor, such that the stack moves by less than 1 mm and then sits there for a moment, before making the next step.

Switch

A very simple method to accomplish this is to insert a switch into one of the power lines going to the motor, and to manually pulse the motor. This also allows for a quick assessment of whether the laminator in question is able to provide the right combination of heat and pressure needed for our purposes.

Warning ! The motor may run at mains voltage. This means that the switch must be rated for this voltage and that it must be properly isolated.

Also bear in mind that the rolls may overheat on the side closest to the heating element and get damaged if stopped too long. Therefore, it may be prudent not to leave the laminator switched on for extended periods of time without regularly activating the transport mechanism.

Relay

A slightly more advanced solution would be to replace the switch with a relay, which can be controlled by an oscillator circuit or a microcontroller. In this case, there should be a manual override to make the motor run continuously, e.g., when feeding or ejecting paper, and when forcing the board between the transport rolls. The circuit can be powered with batteries or with a transformerless power supply.

Warning ! Transformerless power supplies are inherently dangerous to work on. At least an isolation transformer should be used while developing and testing such a circuit.

Triac

Last but not least, if the motors runs on AC power, a thyristor or a triac could be used instead of the relay. This reduces power consumption and allows the motor power to be switched exactly at the zero crossing. The author is presently working on such a circuit.

14.1.4 Results

If used properly, a laminator can yield results of a quality virtually unattainable with manual ironing. Figure 14.5 shows an example of this.

As an added bonus, it may be possible to use a temperature where the transfer paper does not yet begin to stick to the board. This greatly simplifies its removal, and it also yields a very precise transfer, allowing one to spot flaws caused by toner left on the transfer paper immediately.

14.2 The domestic reflow oven

- oven: oven, solder paste, faster, higher precision, handle MLF etc., more costly, multi-lead require rework, previously discussed concepts apply, oven types, aluminium foil

TO DO: **different acid** TO DO: **move SSOP somewhere around here**

14.3 Coating

As an interlude before proceeding with soldering, we may clean the board and cover it with protective coating. This part is experimental and is discussed

Figure 14.6: A small toaster oven can be used to approximate the industrial reflow process.

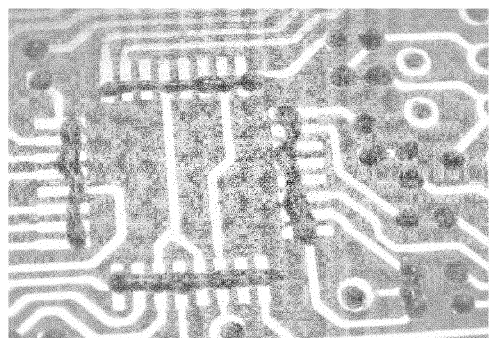

Figure 14.7: ...

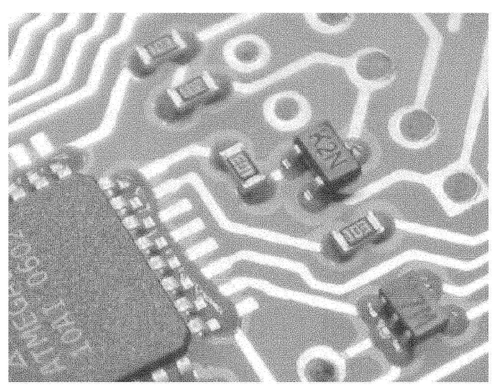

Figure 14.8: ...

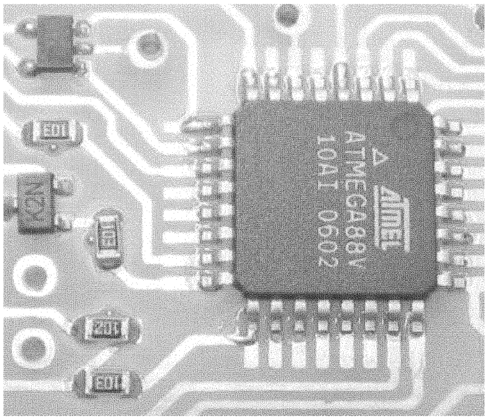

Figure 14.9: ...

below. If you choose to apply no coating, you can proceed with soldering and do the cleanup at the end.

The advantage of coating the circuit is that this protects it from moisture and other external influences that can cause corrosion. To do so, the board must be free from moisture and flux (which is a corrosive substance) before the coating is applied. The disadvantage of coating is that it complicates any repair work that may be needed after the circuit is completed.

TO DO: **mention that tinning would be better** TO DO: **can solder through, but messy** TO DO: **also: electromechanical components have to be soldered after coating** TO DO: **humidity may remain trapped under the coating** 8.3

Coating

The current board production process leaves the board's copper bare and exposed to the forces of nature, allowing it to corrode. This is not only esthetically unsatisfactory, but also bears the risk of eventually causing the device to fail.

As a short-term solution, the board can be coated with an acrylic spray. This has several disadvantages, as described in section 14.3.

A better solution would be to locate the chemicals needed to cover the copper traces with tin, but the author has so far been unable to obtain the necessary materials.

Chapter 15

What's next ?

15.1 Design parameters

- grid, distance - hole diameters - copy silk to copper - minimum space between components - keep inside traces away from pads - trace between SOT-325 pins OK (including diagonal) - avoid soldering top and bottom

15.2 Process improvements

TO DO: **initial programming** TO DO: **tester** TO DO: **clean up the layout** TO DO: **the etching and electronics sections contain too many unrelated items** TO DO: **consistent style – is it "you", "we", or the passive voice ?** TO DO: **motivation**

Figure 15.1: Helping hands, from left to right: flexible arms, regular model with solder iron stand and a sponge receptacle, and a regular model with a magnifying glass. Below, there is a tip cleaner with bronze wool.

Bibliography

[1] Almesberger, Werner. *The LED Toy — User's Manual*, not yet written.

[2] Williams, Tim. *The Circuit Designer's Companion*, Second edition. Elsevier, 2005.

[3] Gootee, Thomas P. *Easy Printed Circuit Board Fabrication — Using Laser Printer Toner Transfer*. `http://www.fullnet.com/u/tomg/gooteepc.htm`

[4] Maxon, Kenneth. *Have you seen my new soldering Iron?* `http://www.seattlerobotics.org/encoder/200006/oven_art.htm`

Appendix A

Shopping references

Table A.1 lists prices and sources (in Buenos Aires) for most of the tools and consumables described in this document. With the exceptions noted below, these items and sources are just examples. Note that availability and prices tend to change with time.

Table A.2 lists items, for which no price could be obtained at the time of writing, or which are sufficiently generic that a large number of equivalent brands can be found at many shops.

There are some product categories where the author has found it difficult to get just the right tool, in terms of quality and in terms of being particularly fit for the purpose of handling small components. Below is a brief description of the problems encountered, and the author's recommendations, after comparing various products.

Angled tweezers Angled tweezers are the most useful form of tweezers for placing components and for gently pushing on them. The ideal tweezers must be small, but still big enough to give a solid mechanical contact. Goot TS-15 are just the right size. Available at Electrocomponentes.

Desoldering wick Desoldering wicks differ greatly in terms of structural integrity and their ability to absorb solder. Goot CP-2015, 2 mm wide, excels in both regards. For the extreme opposite, try Pro's Kit 8PK-031A. Both are available at Electrocomponentes.

Wire cutter The shape of the point of a wire cutter determines how close to an obstacle, e.g., a solder joint, we can cut a wire. This is particularly important when cutting the wires forming vias, where extremely little space is available. The Piergincom TRE-02-NB offers a maximum of precision. Available at Electrocomponentes.

Wire stripper There are many tools for wire stripping, but only few yield dependable results for fine wires. Surprisingly, the very simple and straightforward Pro's Kit PK-3001 (also known as 8PK-3001) gets the job done with amazing precision. There is also the PK-3002 for thicker wires. Lorenzo Tools used to carry this item, but they no longer seem to stock it.

Below are the addresses and Web sites of the shops mentioned above:

DASA Metalúrgica DASA, Montevideo 525, Capital Federal.
`http://www.dasametalurgica.com.ar/contacto.htm`

Digi-Key Digi-Key Corporation, USA. `http://www.digikey.com/`

Easy Easy Argentina.
`http://www.easy.com.ar/`

Electrocomponentes Electrocomponentes S.A., Solís 225, Capital Federal.
`http://www.electrocomponentes.com/`

Item	Price		Shop, product code	Date seen
		— Cutting and drilling —		
Large cutter	9.00	ARS	Lorenzo, AGS0036	10.2006
Small cutter	3.00	ARS	GA-TA	2004
Metal ruler	29.00	ARS	Lorenzo, 46-534	10.2006
Small saw	17.00	ARS	Lorenzo, 1012	10.2006
Cut-off discs	20.00	ARS	Lorenzo, 420	10.2006
Adjustable mandrel	35.00	ARS	Lorenzo, 4486	10.2006
Manual drill	15.00	ARS	Lorenzo, "Mandril Porta mechas manual"	10.2006
Face mask (10 units)	5.00	ARS	Lorenzo, BARB	10.2006
		— Toner transfer and etching —		
Isopropanol, spray	12.00	ARS	MyL, Compitt Prophyl	10.2006
Compressed air	24.00	ARS	MyL, Compitt OR Duster XL	10.2006
Clothes iron	35.00	ARS	Frávega, 25-300-8	10.2006
Ferric chloride, 250 ml	5.50	ARS	MyL, CL1	10.2006
Latex gloves (100 units)	18.00	ARS	Lorenzo, 284	10.2006
Poncho, plastic	10.00	ARS	Lorenzo, 68123X	10.2006
Acetone, 1 liter	9.30	ARS	Retienne	9.2006
		— Soldering —		
Basic soldering iron	15.50	USD+IVA	Electrocomponentes, KX40R	10.2006
Solder 40/60, 0.7 mm	23.00	ARS	MyL	10.2006
Flux, syringe, 10 ccm	5.50	ARS	MyL, Contacflux GEL	10.2006
Flux remover	14.00	ARS	MyL, Removedor de flux	10.2006
Desoldering wick	2.55	USD+IVA	Electrocomponentes, CP2015	10.2006
Precision wire cutter	9.75	USD+IVA	Electrocomponentes, TRE02NE	10.2006
Wire stripper	39.00	ARS	Lorenzo, PK-3001	12.2005
Tweezers, angled	5.87	USD+IVA	Electrocomponentes, TS15	10.2006
Helping hand with magnifying glass				
	20.00	ARS	Lorenzo, "Lupa Auxiliar con Cocodrilos"	10.2006
		— Testing and repair —		
Multimeter with buzzer	16.00	ARS	Lorenzo, ZR-160/JA-830D	10.2006
SMD removal kit	30.00	USD+IVA	Electrocomponentes, SMD1	10.2006
AWG30 wire, isolated	31.42	USD+IVA	Electrocomponentes, W.W.NEGRO/100	10.2006
		— Advanced tools —		
Laminator	247.53	ARS	DASA, ES-400	12.2005
Toaster oven	149.00	ARS	Frávega, TRO210	10.2006
Solder paste	42.00	USD+import	Digi-Key, KE1507-ND	10.2006

Table A.1: Examples for sources and prices for tools and consumables described in this document.

Item	Brand	Shop	Date seen
Dremel rotary tool	—	Easy, Lorenzo	2005
Protective glasses	—	Easy	2005
Steel wool	Virulana, etc.	Supermarket	—
Bronze wool	—	Supermarket	—
Paper towels	—	Supermarket	—
Stapler	—	Office supplies	—
Permanent marker	—	Office supplies	—
Drill bits, 0.8 mm	—	GA-TA	2005
Water basin	—	Supermarket	—
Etching tray	—	Supermarket	—

Table A.2: Generic items and products for which no recent price information was available at the time of writing.

Frávega Frávega S.A.
`http://www.fravega.com.ar/`

GA-TA GA-TA S.A., Tte. Gral. Perón 1314, Capital Federal.
`http://www.gatatornillos.com.ar`

Lorenzo Lorenzo Tools, Av. Corrientes 2621, piso 2, oficina 22, Capital Federal.
`http://www.lorenzotools.com/`

MyL MyL Transformadores SA, Paraná 229, Capital Federal.
`http://www.myltransformadores.com.ar/`

Retienne Eduardo Retienne S.A., Tte. Gral. Perón 1155, Capital Federal.

The author is a regular and generally satisfied customer of the companies mentioned in this appendix, but has no interest in them, financial or otherwise, beyond this.

Appendix B

Terminology in Spanish

This appendix contains translations for the names of various technical items mentioned in this document. The translations are mainly intended for shopping purposes, i.e., to provide the term under which a given tool or material is known in the Spanish-speaking world in general, and in Argentina in particular.

acetone acetona

collet pinza de sujecion, portabrocas

compressed air aire comprimido

cut-off disc disco cortador / de corte

desoldering braid / wick cinta para desoldar, malla desoldante

double-sided doble faz

dremel minitorno

drill bit mecha, broca

face mask barbijo

ferric chloride cloruro férrico

flux remover removedor de flux

helping hand tercera mano

iron (*for clothes*) plancha

isopropanol alcohol isopropilico

laminator plastificadora

latex gloves guantes de latex

linen lino

mandrel mandril, portabrocas

multimeter multimetro, tester

paper towel rollo de cocina

pincers alicate, pinza

PCB placa (de circuito impresso)

printed circuit board placa (de circuito impresso)

ruler regla, (*in this context*) escuadra

saw (arco de) sierra

single-sided single faz

solder estaño

solder paste pasta de soldadura

soldering iron soldador

steel wool lana de acero, virulana

tip (*of a soldering iron*) punta

tweezers brusela, pinza

wire stripper pela cable

wool lana