International Journal of Engineering Innovation & Research
Volume 2, Issue 2, ISSN: 2277 – 5668

Experimental Analysis of Small Scale PCB Manufacturing Techniques for Fablabs

Yannick Verbelen
Erasmus University College Brussels,
Nijverheidskaai 170,
1070 Anderlecht, Belgium
yannick.verbelen@ehb.be

Davy Van Belle
Erasmus University College Brussels,
Nijverheidskaai 170,
1070 Anderlecht, Belgium
davy.van.belle@gmail.com

Jelmer Tiete
Erasmus University College Brussels,
Nijverheidskaai 170,
1070 Anderlecht, Belgium
jelmer.tiete@etro.vub.ac.be

***Abstract* – In this paper we present a complete modular PCB manufacturing process on fablab scale that is compliant with current PCB manufacturing standards. This includes, but is not limited to, a minimum track width of 8 mil, a minimum clearance of 6 mil, plated and non plated holes, a solder resist, surface finish and component overlay. We modularize industrial manufacturing processes and discuss advantages and disadvantages of production techniques for every phase. We then proceed to discuss the relevance and added value of every phase in the manufacturing process and their usefulness in a fablab context. Production techniques are evaluated regarding complexity, overhead, safety, required time, and environmental concerns. To ensure practical feasibility of the presented techniques, the manufacturing process is benchmarked in FablabXL and aims to be a practical reference for implementing or extending PCB manufacturing activities in fablabs.**

***Keywords* – Printed Circuit Board, Fablab, Standard Pool, Manufacturing.**

I. INTRODUCTION

Prototyping circuits is an essential phase in any electronic design process. Prototypes of electronic circuits have traditionally been built on turret boards or breadboards [13, 14], the latter remains the most common prototyping technique to date. Many high frequency designs however cannot be prototyped effectively on breadboards because of their excessive parasitic capacitances between adjacent columns and high inductances in the connecting wires. In recent years, rising popularity of surface mount technology (SMT) has reduced the availability of classic through hole packages such as DIP in favor of leadless equivalents. Where both through hole and SMT packages are available for a part, the through hole variants are typically harder to acquire and more expensive [12, 15]. New designs are often only available in SMT packages, and industry evolutions anticipate on through hole packages becoming obsolete by 2018 or earlier. Leading semiconductor manufacturers have pointed out that DIP packages for classic series such as 40xx and 74xx are only kept in production to provide replacements for legacy electronics and to allow breadboard prototyping for educational purposes. Also classic discrete components such as the BF245 are discontinued in TO-92 package [8].

The mentioned industry evolutions have increased the importance of printed circuit boards (PCB) in electronics prototyping. Unlike with breadboards, parasitic properties of PCBs are precisely known [11]. This allows designers to minimize the influence of parasitic resistances, inductances and capacitances on the circuit's behavior by using matching networks where necessary [6, 7]. PCBs are also ideally suited for prototyping of SMT parts, either by direct prototyping or by means of intermediate SMT to through hole breakout boards [16]. Furthermore, PCBs have the additional advantage of providing robustness that breadboards cannot achieve. Finally, PCBs can be duplicated with ease, allowing parallel testing of multiple (nearly) identical prototypes. Despite their numerous advantages in the prototyping phase of electronic design processes, usage of PCBs outside professional environments and high end academic research groups is undervalued. With the recent introduction of professional yet easy to use and affordable design suites [10], the design of a PCB is no longer a prohibitive factor. In contrast, production processes for PCBs have become more complex due to environmental limitations and advancing electrical requirements. An immediate consequence is that PCBs are getting more expensive and take longer to manufacture. Precisely these properties are fatal in an educational context or prototyping environment such as a fablab.

Several suppliers offer equipment for in house prototyping of PCBs, specifically marketed to fablabs, universities, start-ups and other customers demanding low prototype quantities. Due to the large number of production steps, the total cost of all the equipment needed to manufacture a single board can be 3 to 4 orders of magnitude higher than commercial manufacturing, and the time required can range from a few hours to as much as an entire day. In these circumstances it is often forgotten that alternatives do exist and can provide the fast and low cost PCBs fablabs require when properly combined together. Hobbyists in particular have contributed significantly to small scale PCB production technology in recent years [2]. In this paper, section II starts with an overview of commercial PCB manufacturing processes. In section III we break down the process and provide solutions for every production phase, with a discussion of advantages and disadvantages. Section IV compares the achieved results with existing technology, and section V presents some suggestions for future optimizations. Finally, section VI concludes the paper with a summary of the presented work.

II. COMMERCIAL PCB MANUFACTURING PROCESSES

Commercial PCB manufacturing facilities start from a substrate that is coated either single sided or double sided with a copper layer, called a copper clad board. Copper clad boards are typically 1.6 mm thick, with an FR4 composite substrate and copper layers of 35 µm in thickness. Other substrate or copper layer thicknesses or substrate materials are also commercially available but dedicated to specific applications that are rarely prototyped. The techniques described hereafter are targeted to 1.6 mm single or double sided FR4 boards, but it should be noted that other copper clad board types can also be processed with minimal adjustments [1]. Aside from the copper clad boards themselves, the PCB manufacturing process requires numerous chemicals for coating, laminating, etching, developing and washing boards as they progress through the different manufacturing phases.

The set of features that is commonly regarded as standard for general purpose PCBs is provided by most commercial PCB manufacturers in the standard pool [3]. Said features include single and double sided 35 µm copper layers on 1.6 mm thick substrates, plated and unplated holes, tented and untented vias, 0.3 mm minimum hole size, 8 mil minimum clearance and track width, green solder mask on all copper coated outer layers, copper surface finish and silkscreen component print on top side. These are also the minimum requirements that should be aimed for when conceiving an in house PCB manufacturing facility.

III. BREAKDOWN OF THE PCB PRODUCTION PROCESS

The PCB manufacturing process as discussed below is split up in 7 phases, each consisting of one or more steps. The manufacturing process can be halted after every phase, allowing a trade off of speed for quality. The 7 phases can be seen in figure 1 and include the Copper Routing Phase (A), the Drilling Phase (B), the Solder Resist Phase (C), the Though Hole Plating Phase (D), the Surface Finish Phase (E) and finally the Component Overlay Phase (F) . Each phase will be discussed separately below.

A. Copper Routing Phase

The Copper Routing Phase is the only essential phase of the manufacturing process. Routed PCBs have their copper artwork transferred, but no holes or finishes, hence their use is almost exclusively limited to SMT breakout boards.

Two major methods exist for routing copper clad boards: mechanically through milling, and chemically through etching.

Although several suppliers offer PCB milling machines that are capable of milling tracks in the copper clad board, drill holes and route board cutouts, the disadvantages outnumber the advantages. Firstly, the purchase of a PCB milling machine or any other generic milling machine with comparable speed and accuracy presents a prohibitive cost for fablabs. Prices range from ca. € 5 k to well over € 40 k for advanced models with high accuracy and automatic tool changers. Secondly, PCB milling machines require a processing time that is proportional to the complexity of the board(s) being milled. For large and complex boards, milling times of several hours are not exceptional. Models without automatic tool changers also require an operator to be nearby the machine to exchange tools or replace broken or blunt tools. The only three significant advantages provided by PCB milling in comparison to etching are that milling is more environment friendly due to the absence of chemicals and the easier reuse of removed copper, the automatic drilling of holes with high accuracy, and better performance for RF PCBs.

Efforts are put into the development of a new generation of PCB milling machines. Notable examples are LPKF developing faster laser milling machines that do not require mechanical tools, and Elektor providing a more affordable PCB milling machine for less than € 5 k. Neither of these can provide an effective solution for fablab prototyping, however.

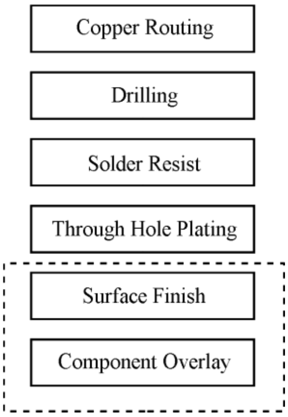

Fig.1. Overview of the PCB manufacturing process. Every phase can be further divided in smaller steps. The Surface Finish and Component Overlay phases are optional for prototypes.

The alternative to milling is etching, which refers to chemically removing undesired copper from the copper clad board by means of a corrosive chemical etchant. Depending on the used etchant type, concentration and temperature, etching times range from a few seconds to dozens of minutes. The time required is independent of board size or board complexity, making it specifically suited for high volumes and parallel etching. Copper routing using chemical etching requires 4 steps: transferring the artwork to an etch resist, developing the resist, etching the board, and finally stripping the etc resist from the copper.

1) Transferring artwork to etch resist

When etching a copper clad board, the board is entirely submerged in etchant. To protect desired copper areas such as tracks and polygons from the etchant, a coating that is able to resist the etchant must be applied prior to etching.

This photosensitive coating can either be applied to a blank board as a film, or prelaminated boards can be readily purchased. Chemical through hole plating requires lamination of a photosensitive coating after the plating process, and that is the only significant reason to choose for copper clad boards instead of prelaminated boards. Copper clad boards and prelaminated boards are currently comparable in price for medium volumes, and since an easier alternative for chemical through hole plating is suggested below, the choice for prelaminated boards was made.

To transfer the copper layer information to the board, it is first printed on cellulose acetate transparencies using a high resolution laser printer. The printing quality of the printer determines the accuracy of the final board, e.g. tolerances, minimum width and minimum clearance. The contrast of the printed tracks must be sufficiently high, and the aspect ratio must be within acceptable tolerances (e.g. the image may not be stretched in either direction because of thermal distortion of the plastic). This accuracy also affects solder resist and component overlay, as discussed in the next paragraphs.

Once printed, the protective layer is removed from the prelaminated board and placed in an ultraviolet (UV) exposure unit. The transparency is then aligned with the board, and exposed to ultraviolet light to cure the uncovered parts of the photosensitive laminate. Tracks are covered by ink on the transparency and protected from ultraviolet light. Any ultraviolet light source can be used to cure the photosensitive coating from a technical perspective, including sunlight. It should be noted that ultraviolet light poses several health hazards, however, including adverse effects on skin and eyes. Consequently, we consider an integrated UV exposure unit to be mandatory, but this can be built with relative ease in a modern fablab. As light source, phosphorescent tubes and LEDs provide comparable performance, but LEDs are currently more expensive in the required frequency range. To achieve an 8 mil accuracy for track widths, an integrated vacuum unit is also required to press the transparency on the board surface and prevent any ultraviolet light from getting between the ink and the board.

If a double sided board is desired, a minimum of 3 assymmetric holes must be drilled through fiducial markings to allow transparents for top and bottom side to be aligned properly. For this purpose, we developed a custom fiducial marking as footprint with a 1 mm through hole in the center and concentric circles on all copper layers, solder resist layers and component overlay layers.

2) Developing the etch resist

After curing the photosensitive coating above undesired copper, it must be washed off before commencing the etching process. This is done by developing the photosensitive coating in a 1 % sodium hydroxide (NaOH) solution at room temperature. The NaOH concentration is critical: if it is too low, the board is insufficiently developed and will etch incompletely. If the concentration is too high, also the uncured photosensitive coating will be washed off, resulting in a blank copper clad board. The NaOH does not need to be pure for this purpose. Our experiments have shown that NaOH solutions prepared using NaOH pellets sold as drain cleaner under various brands do not result in lower performance, but are considerably cheaper. After developing, the board is rinsed with water to prevent NaOH from entering the etching solution.

3) Etching the board

Any chemical compound or combination of compounds that dissolves metallic copper (Cu) but does not harm the etch resist on the board can be used as etchant. Etchants are generally divided in 5 groups: peroxides, metal chlorides, chlorine based etchants, persulfates and nitrates. The most common etchants for PCB manufacturing are discussed below.

1. *Ferric Chloride* ($FeCl_3$) is one of the most commonly used etchants for PCBs because it is relatively inexpensive and etches quickly. It is commercially available as pellets or as a ready to use solution. Its drawbacks are significant, however: used etchant poses an environmental hazard ($2\,FeCl_3 + Cu \quad 2\,FeCl_2 + CuCl_2$) and makes stains that are hard to remove from equipment and clothing unless dissolved in organic acids such as oxalic acid. Another practical problem with ferric chloride is the dark color of the liquid itself, which prevents visual inspection of the etching process and requires the board to be removed from the etch tank and rinsed before it can be checked. In summary, the use of ferric chloride as etchant is effective but messy.
2. *Hydrochloric acid and hydrogen peroxide* (HCl and H_2O_2) is an industrial etchant and more than an order of magnitude faster than ferric chloride. It is able to etch a board in less than 10 seconds, depending on the concentration. A notable advantage is that the reaction proceeds well at room temperature, eliminating the need for a heated etch tank. Although the reaction does not produce explosive hydrogen gas ($H_2O_2 + 2\,HCl + Cu \quad CuCl_2 + 2\,H_2O$), the exothermic reaction causes the acid to heat up and release hydrochloric gas. This gas is harmful when inhaled, and corrodes any nearby metallic object. In addition, when this echant is accidentally mixed with acetone in the next cleaning step, the highly unstable and explosive compound acetone peroxide may be formed. Unlike ferric chloride, the combination of hydrochloric acid and hydrogen peroxide is unstable. The etchant decays in storage, reducing its shelf life. Since expired etchant does contain the highly toxic $CuCl_2$, it is unsuitable for low volume etching despite being the most inexpensive of the discussed etchants. Prohibitive in this context is the need for a fume hood for indoor use, and the many physical hazards associated with this etchant.
3. *Sodium persulfate* ($Na_2S_2O_8$) has only recently gained attention as etchant because it is a relatively expensive etchant and etches slowly. This last property can also be regarded as an advantage since it allows

more precise control over the etching process, and for this reason the etchant has become commercially available under the name *fine etch crystals*. Unlike ferric chloride, sodium persulfate is not hygroscopic, which allows it to be stored for prolonged periods of time in dry state. Once dissolved in water it becomes unstable, with decay times depending on the temperature of the solution. The optimal temperature for etching with sodium persulfate is between 40 and 50, which requires an etch tank with heater. An important advantage of sodium persulfate in solution is that it appears as a water clear liquid that slowly turns blue as the reaction progresses ($Na_2S_2O_8$ + Cu Na_2SO_4 + $CuSO_4$). At any time the solution remains transparent and does not produce any gasses or fumes. This makes it convenient to follow the process and remove the board at the exact right moment from the solution. In contrast with $CuCl_2$ which is highly toxic, the reaction of sodium persulfate with metallic copper produces harmless sodium sulfate and moderately toxic copper sulfate. This makes disposal of used etchant less of an environmental problem. The ammonium equivalent $(NH_4)\ _2S_2O_8$ can also be used as etchant, but sodium had the advantage that no chemical reaction occurs when it is accidentally mixed with the other chemicals in the process, NaOH and Na_2CO_3.

It should be noted that some etchants are unsuitable because these are either unstable (hydrogen peroxide and sulfuric acid, hydrogen peroxide and sodium bisulfate, ...), emit hazardous gasses (sodium hypochlorite and hydrochloric acid, manganese dioxide and hydrochloric acid, ...) or are highly corrosive (nitric acid, potassium nitrate and hydrochloric acid, ...). For these reasons, these alternative etchants will not further be discussed.

After testing the three etchants mentioned above, we found sodium persulfate to be most suited because of the high accuracy it allows. The solution must be heated however, requiring an etch tank. We found that a concentration of 200 g/L at a temperature of 45 is ideal. Etching requires ca. 15 minutes with these parameters, depending on whether immersion or spray etching is used. The hydrogen chloride and hydrogen peroxide mixture can be suitable for low complexity boards because it is faster and does not require an elevated temperature or bubble circulation system, but the caustic and toxic fumes require a fume hood or other type of ventilation.

4) *Stripping the etch resist*

After the etching process is completed, the etch resist must be removed to expose the copper tracks on the board so they can be treated with a surface finish in phase E. Soldering a board with etch resist on is not possible either.

To remove the etch resist, the board is first washed with acetone to dissolve the coating, and then rinsed with water to remove the contaminated acetone.

B. *Drilling Phase*

The second phase after etching the board is drilling its holes. This can be done either manually or with a CNC drilling machine. Experiments show that for boards with few holes, the setup effort of a CNC drilling machine takes longer than drilling the board manually. Automatic drilling also tends to be inefficient for boards with a large number of drill sizes, since tool exchange on CNC drilling machines takes longer than on a manual drill press.

Unless a high accuracy is required for hole positions, we conclude that in academic contexts a manual drill press is more efficient and a more cost effective investment than a CNC drilling machine.

It should be noted that drill sizes need to be adjusted for through hole plating, as will be discussed below.

C. *Solder Resist Phase*

Applying a solder resist to the board is essential to protect the copper against corrosion, to isolate components with conductive undersides from the tracks running beneath them, and to prevent solder bridges during soldering. Despite its advantages, academic and fablab PCBs are typically not fitted with a solder resist layer because it is thought to be a complicated and messy process.

On an industrial scale, solder resist is commonly applied by silkscreen printing or by curtain painting and hardening a layer of solder resist and curing it with ultraviolet light.
Silkscreen printing is a useful method once the screen has been prepared because it allows easy duplication of a design. The preparation of the screen is time consuming however, making it less suited for prototypes.

To reduce the amount of chemicals used in the manufacturing process for safety, health and environmental reasons, we found that the best way to equip prototype boards with a solder resist is to use off the shelf solder resist laminate. This laminate consists of a layer of solder resist and adhesive sandwiched between 2 protective plastic films.

Application of solder resist laminate

The solder resist laminate is commercially available in rolls of 1 to 5 m with a width of ca. 30 cm. The laminate is sensitive to ultraviolet light, so it must be stored in a dark container. After cutting a piece of laminate the size of the board, the protective film on the underside of the laminate is removed to expose its adhesive side. The laminate is then applied to the board using a standard office laminator. We found that cold lamination gives better results than hot lamination because hot lamination risks damaging the integrity of the fragile solder resist film. The laminate attaches better to copper than to blank FR4, which is a concern for boards with large areas without tracks, polygons or fills.

1) *Transferring atrwork to solder resist*

The process for transferring the artwork to the solder resist is identical as in the copper routing phase A, but with a different exposure time. We found that an exposure time of 30 s for a 90 W ultraviolet exposure unit gives the best resolution. Accuracies of 10 mil are achievable, but proper alignation of the solder mask with the copper layer is usually the limiting factor. A glass plate can increase the accuracy with which both layers are aligned when placed on a light table. Board and solder mask are then attached to the plate using adhesive tape, and the whole assembly is placed in the UV exposure unit.

After exposing the solder resist, it should be allowed a minimum of 5 minutes in the dark to complete polymerization.

2) Developing solder resist

After exposure and cool down of the PCB, the unexposed solder resist must be removed. This is done in a 1 % sodium carbonate (Na_2CO_3) negative developer bath, which dissolves the unexposed solder resist film. For general purpose boards, bath temperature and Na_2CO_3 concentration are not critical parameters. The unexposed film will detach from the board surface and can be washed off with a paintbrush after ca. 90 s. Another 60 s later the PCB is removed from the bath and rinsed with water to clean off Na_2CO_3 remains and allow final cleaning of exposed copper with a brush.

We observed deterioration of the resist around the pads if the board is left in the developer for longer than 5 minutes. This is caused by the Na_2CO_3 dissolving the adhesive layer from the sides, and results in the solder resist around pads peeling off from the board. Due to the water clear nature of the Na_2CO_3 solution, this situation can be prevented by continuously inspecting the development state of the uncured solder resist film.

IV. THROUGH HOLE PLATING PHASE

Plating holes is a mandatory step in the production process of most double sided PCBs with vias to connect copper on both sides. During the plating phase, the inner surface of holes is coated with copper to conduct current between the connecting layers. As with the other productions steps, conventional PCB manufacturing employs a chemical electroplating process to apply this conductive coating. Despite efficient on an industrial scale, the method has several drawbacks on laboratory scale:

- Special equipment is required to achieve a uniform thickness of the coating in all holes.
- The various chemical processes use numerous corrosive, toxic and environmentally harmful chemicals.
- The complete procedure requires as much as 90 minutes on laboratory scale.
- Successful plating of holes < 0.35 mm is not guaranteed.
- Etch resist for copper routing must be laminated over the through hole plated board in an extra step.

Chemical through hole plating also has financial drawbacks. The limited availability of activator chemicals in small volumes makes them expensive. These chemicals also decay within days once mixed, resulting in a large amount of wasted chemicals for low volumes.

Copper rivets can provide a suitable alternative to electrochemical through hole plating. Standard PCB rivets are available for a board thickness of 1.6 mm in various diameters. A rivet press is required for insertion of the rivets, and select vendors offer specialized PCB rivet presses. The rivet press plastically alters the shape of the rivet during insertion, locking it into place, and making removal impossible. A satisfactory conduction can be achieved only when the rivet is properly inserted.

Using rivets for through hole plating is considerably cheaper than electrochemical through hole plating because the rivets are the only consumables. An investment of around €300 for a rivet press must be made, and rivets cost between 2 and 3 cents per piece.

Inserting rivets requires some practicing, and a speed of 5-10 rivets per minute is achievable. Small rivets typically take longer to insert. At this rate, boards up to mediocre complexity can be plated an order of magnitude faster than with electrochemical through hole plating.

We experienced two drawbacks of rivets. Firstly, only holes with a discrete number of diameters can be plated. Most vendors supply between 4 and 8 rivet diameters, typically from 0.3 mm up to 2 mm. Other diameters need to be rounded to the nearest larger diameter. Secondly, it should be noted that tool bits for small diameters tend to bend and break very easily, particularly in the hands of inexperienced students or fablab visitors. Replacing broken tool bits must be calculated in as a significant expense in early months of operation.

Given the disadvantages of electrochemical through hole plating regarding cost, environmental safety and chemical hazards for students, we believe rivets provide a superior technology for fablab scale through hole plating. The technology is safe, clean, does not produce any waste, and is cheaper than electrochemical through hole plating. In addition, the operator can easily make the difference between plated and non plated holes.

For complex SMD boards with vias underneath SMD ICs, it is possible to make tented vias using the technique described above. Vias that must be tented can be riveted before applying the solder resist.

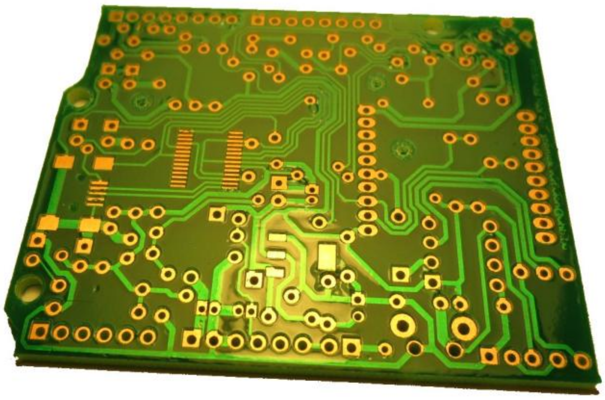

Fig.2. Single sided custom designed and manufactured Arduino Uno Shield with through-hole components and surface mount (SMT) components ready for soldering. A surface finish has not been applied.

Although this works for vias that have a sufficiently flat top side, the top copper ring of thicker vias may cause a deformation of the solder resist under excessive pressure of the laminating machine. With this potential problem in mind, one should carefully consider whether or not a via must be tented.

V. SURFACE FINISH PHASE

The board can be considered finished after the through hole plating phase: the board can be soldered manually or automatically (reflow) and has the expected electrical and mechanical characteristics. In our experience this is sufficient for fablab prototypes and all but a few student projects.

When desired, two enhancements can further improve the performance of the finished boards: a surface finish and a component overlay. The first enhancement to be discussed is the *surface finish*.

After applying solder resist to the board, any copper area uncovered by the solder resist is exposed to oxygen in the air. This will cause pads and vias to start oxidizing slowly over time. Fingerprints are also capable of oxidizing the copper in minutes. The resulting layer of copper oxides obstructs heat conduction to the board during soldering, making manual soldering harder and increasing the risk of tombstoning during reflow soldering. The electrical resistance of the solder joints will also be higher than predicted because copper oxides are electrical insulators.

When the board is not intended to be soldered immediately after manufacturing, it is advisable to apply a surface finish to exposed copper area. Various methods exist for the application of protective coatings, including SMOBC, HASL, SN100C, HAL, Immersion Tin and Immersion Gold. Many types of surface finish are no longer permitted due to RoHS restrictions. The best surface finish known to industry is Immersion Gold. Boards finished with Immersion Gold have a perfectly even surface and excellent solderability combined with a long shelf life. Although Immersion Gold is the most expensive surface finish, it is quickly growing to the industry standard because boards treated with this finish are compliant with all environmental regulations.

After careful comparison of the cost, electrical characteristics and application processes of different board finish types, we conclude that Immersion Gold is not the best choice for fablab PCBs. Immersion Tin can provide a cheaper and easier alternative because tin can be applied to the board surface by a simple ion displacement reaction with the top copper layer. Some surface finishes require electrochemical processes comparable to electrochemical through hole plating, but non-electric methods are also widely available and referred to as ’cold tinning’.

Any compound that releases tin in the solution can be used, but for most compounds the reaction speed is too slow to be of practical use. A solution of 3.8 g/L $SnCl_2$, 49.5 g/L $SC(NH_2)_2$ and 12 mL 98 % H_2SO_4 in demineralized water is described in metallurgic literature [5]. Reaction speed is satisfactory at room temperature despite an optimal reaction temperature between 25°C and 50°C [9].

A problem with all examined cold tinning formulae is their toxicity. This includes the tin(II)chloride formula above, which includes the carcinogenic, toxic and environmentally harmful compound thiourea (LD_{50}125 mg/kg for rats). Handling the cold tinning solution requires strict safety precautions including gloves, mouth mask and fume hood, and is as such not suitable for use in a fablab.

When a surface finish is required for a specific application, we recommend using off-the-shelf solutions using the thiourea formula, commercially available under brand names *Seno* and *Sur Tin*, instead of preparing the solution oneself. Regardless, even working with these solutions involves severe health risks, and consequently we do not recommend the application of a surface finish unless it is strictly necessary.

Evidence exists for wiskering of boards with this tin surface finish applied [4]. So far we have not observed this phenomenon however, and do not consider it a major problem for the tin surface finish.

VI. COMPONENT OVERLAY PHASE

Equipping the PCB with a component overlay is a last possible enhancement. It makes it easier to populate with components without the original design files, and serves as a method of adding version information, project name etc. to the board.

The industrial method for adding component overlay to the PCB is through *silkscreen printing*. Although feasible on a small scale, the efforts required to screen print a component overlay on the PCB outnumber its advantages. Also for this process, better alternatives exist.

We found that 2 techniques for adding a component overlay can compete, and whichever is preferred is dependent of the type of board.

For single sided boards, having a blank top layer without copper or solder resist, the fastest and cheapest technique is by *toner transfer*. This can be most easily done by printing the component overlay mask mirrored on paper using a laser printer, and then transferring the artwork to the board with an iron. For proper alignment of the artwork with the holes on the board, fiducial markings are mandatory. We found that 1 mm holes drilled in the paper in the center of fiducial markings allows precise visual alignment with offsets smaller than 0.05 mm. It should be noted that the component overlay for single sided boards cannot overlap pads, hence exact alignment is not critical. A multicolor component overlay can be created in the same step by using a color printer, which is complex using conventional screen printing technology.

For double sided boards, or boards with a solder resist on both sides, it is not possible to use the toner transfer method to apply a component overlay because the toner does not transfer well to the solder resist. The component overlay can be laminated instead using the same technique as to apply the solder resist but with another color of laminate. A supplier of PCB manufacturing equipment provides a specialized blue laminate dedicated for this purpose. Although a better resolution can be achieved with the laminate method than with the toner transfer method, it is significantly more expensive because the component overlay mask must be printed on a transparency first and because of the laminate cost. Fortunately, Na_2CO_3 developer, laminator and UV exposure unit can be shared with the solder resist phase.

Proper alignment is critical when applying a component overlay to a board side with exposed copper pads, as these may not overlap. This can be accomplished by including the necessary fiducial markings on the component layer in the PCB design to allow correct alignment using a light table (see paragraph II.A.1).

At the moment of publication, no method to apply a white component overlay to a PCB other than silkscreen printing is known to the author.

Phase	Investment	Eurocard Cost	Time
Copper Routing	€ 1,300[1]	€ 3.50	20 min
Drilling	€ 500	€ 1.00	5 min
Solder Resist	€ 50[2]	€ 0.75	10 min
Plating	€ 300	€ 0.50	10 min
Surface Finish	0	€ 1.50	5 min
Comp. Overlay	€ 50[2]	€ 1.25	10 min
Total	€ 2,200	€ 8.50	60 min

Table 1: Overview of required investments and costs for production of a single sided Eurocard (160 cm) per phase. Investment cost is averaged and assumes equipment is purchased rather than manufactured in the fablab itself. [1]Assuming a high quality laser printer is already present. [2]Assuming the UV exposure unit is shared with Copper Routing Phase.

VII. COMPARISON WITH COMMERCIAL PCB MANUFACTURING

Table 1 shows an overview of the finances and time involved in the presented small scale manufacturing process. To calculate the required investment, it was assumed that all equipment is purchased from leading high quality manufacturers. It should be noted however that various machines can be built in a well equipped fablab without much effort. The equipment to be purchased includes a UV exposure unit, bubble or spray etch tank with heater, table drill press, laminator, rivet press with tool bits and a clothing iron for a total maximum of € 2,200. A double sided UV exposure unit, valued close to € 1,000 represents most of this investment cost, but can be shared between Copper Routing Phase, Solder Resist Phase and Component Overlay Phase..

In the second column, a breakdown of the costs for the production of one single sided Eurocard, with a size of 10 x 16 cm, is given. As on industry scale, the largest share in the price per board is from the photo sensitized copper clad board itself. The total cost of € 8.50 includes costs for copper clad board and consumables such as rivets, laminates, drill and rivet bit wear, transparencies and chemicals. Water and electricity energy costs are excluded. The exact amount of consumables per step depends strongly on the board design in some phases: boards without polygons or fills will require more etchant, boards with more holes need more rivets etc. Determining the exact cost of a board may be challenging because of these factors, and the values in table I reflect material costs for an average board.

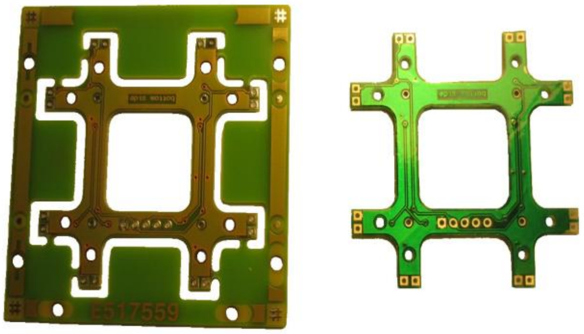

Fig.3. Double sided board (bottom side view) showing a comparison between a commercially manufactured PCB (left) and a fablab scale manufactured PCB using the process described above (right). Tented vias have been used. The board measures 2 x 2 inches.

VIII. FUTURE WORK

Despite the capability to manufacture high quality PCBs on a university or fablab scale, several production steps can be further upgraded.

Firstly, we think a more robust through hole plating technique is necessary. Although the presented rivet method provides highly reliable through hole contacts for a low price, the fragile rivet press tools make it a costly process because they tend to break easily. The care with which rivets need to be inserted also slows down the process considerably. A less error prone insertion method for the rivets in the press would be a considerable upgrade to the through hole plating phase.

Secondly, more research for chemical cold tinning could allow a full integration of surface finish application into the manufacturing process. Current formulae are too hazardous to be handled by non chemists, too expensive and too hard to dispose of after use because of the environmental concerns associated with them. A cold tinning solution that can be stored in a simple container, applied without fume hood, is not hazardous upon contact with skin, and can be safely disposed of is required in a fablab context. Experiments with silver and zinc surface finish solutions are currently in progress.

Third, a method to apply a white component overlay that is easier than silkscreen printing would contribute to a better visual appeal of the finished boards.

Finally, a method to manufacture surface mount solder paste stencils without the need for expensive equipment or consumables is necessary for facilitating the assembly of today’s surface mount boards. Experiments are currently ongoing to develop a photographic method using thicker laminates in contrast with the existing sheet metal stencils which require laser cutters and are more expensive.

IX. CONCLUSION

In this paper we presented a low cost, visual and fast manufacturing process for printed circuit boards (PCBs). The presented process allows high quality PCBs to be manufactured within an hour without the need for expensive equipment or infrastructure. The chemical processes make use of chemicals that do not require a chemist training for safe handling, are widely commercially available, inexpensive, and can be safely disposed of after use.

The presented process was optimized for fablabs, allowing low volume PCB manufacturing with a minimal investment of €2.2k and inexpensive consumables.

ACKNOWLEDGMENT

The authors would like to thank FablabXL for funding the research and providing necessary infrastructure.

REFERENCES

[1] COONROD John, *Understanding When To Use FR-4 Or High Frequency Laminates*, OnBoard Technology September 2011, pp. 30, September 2011.

[2] DANIELS Scott, *Making Your Own Printed Circuit Board - An Overview*, Provide Your Own, April 2012.

[3] Eurocircuits, *Eurocircuits Design Guidelines*, Available: http://www.eurocircuits.com, January 2010.

[4] GALYON George T., *A History of Tin Wisker theory: 1946 to 2004*, SMTAI International Conference, 2004.

[5] Hall N., *Immersion Plating*, Metal Finishing Guidebook and Directory, Metals and Plastics Publications, Inc., pp. 393-396, 1986.

[6] JUNG Walter G., *Op Amp Applications*, Analog Devices, ISBN 0-916550-26-5, Chapter 7, 2002.

[7] KESTER Walt, *High Speed System Applications*, Analog Devices, ISBN-10: 1-56619-909-3, ISBN-13: 978-1-56619-909-4, Part 4, 2006.

[8] KOZLAY Ivan, *BF245 - the legend is leaving, the successor comes*, SOS Electronics, Available: http://www.soselectronic.com/?str=1184, October 2012.

[9] KUNCES David, *Chemical Deposition of Metallic Films From Aqueous Solutions*, William Andrew Publishing, 1990.

[10] MAXFIELD Clive, *Altium redefines board-level design*, EE Times, February 2011.

[11] MUMBY Stephen J., YUAN Jih, *Dielectric properties of FR-4 laminates as a function of thickness and the electrical frequency of the measurement*, Journal of Electronic Materials, Vol. 18, Issue 2, pp 287-292, March 1989.

[12] PEASE Robert A., *Troubleshooting Analog Circuits*, Butterworth-Heinemann, ISBN 0-7506-9184-0, 1991.

[13] RAKO Paul, *Prototyping Techniques: Things to Know Before Pulling the Trigger*, EDN, May 2008.

[14] VERBELEN Yannick, *Geïntegreerd Practicum: Robocup Project*, unpublished, 2009.

[15] WILLIAMS, Jim, *The Art and Science of Analog circuit Design*, Butterworth-Heinemann, ISBN-10: 0750695056, ISBN-13: 978-0750695053, 1995.

[16] ZUMBAHLEN H., *Linear Circuit Design Handbook*, Analog Devices, ISBN 9-780-750687-03-4, April 2008.

AUTHOR'S PROFILE

Yannick Verbelen
graduated M.Sc in industrial engineering with specialization in electronics. He is associated with the Erasmus University College in Brussels and the Vrije Universiteit Brussel. He teaches courses related to electronics, research methodology, statistics and mechatronics while preparing a Ph.D. in energy harvesting and renewable energy. He is responsible for electronics development in FablabXL and expansion of its PCB manufacturing facilities.

Davy Van Belle
was born on March 24, 1986 and is with the Erasmus University College Brussels where he prepares a M.Sc. in Applied Engineering with specialization in Embedded Systems. He has been experimenting with PCB manufacturing techniques for many years with a particular focus on low cost materials to be applied in small batch productions. His current research focuses on photovoltaics for embedded systems. As Research Associate in FablabXL he is responsible for the introduction of electronic innovation and experimental analysis of state of the art technologies.

Jelmer Tiete
is a Ph.D .candidate in the Engineering Sciences at the Vrije Universiteit Brussel and the Erasmus University College Brussels., where he received his Masters in Industrial Engineering. His main research topics cover the fields of wireless sensor networks, sound measurement, and noise monitoring. He is currently developing an environmental monitoring network based on wireless sensor networks. He is interested in electronic prototyping, PCB manufacturing, 3D prototyping, and product development. For this reason he helped start the FablabXL at the Erasmus University College Brussels.