Surface-mount device prototyping in education

A feasible alternative to conventional through-hole practice

Vassilis K. Papanikolaou
Civil Engineering Department, School of Engineering
Aristotle University of Thessaloniki
Thessaloniki, Greece
billy@civil.auth.gr

***Abstract*—A feasibility study is herein attempted, towards the adaptation of modern surface-mount device (SMD) prototyping practice to learning environments. This necessity emerges not only from the profound advantages of the above technology (e.g. component size, availability, low cost etc.) but also from the fact that contemporary designs often require special board layout considerations, which may be incompatible with through-hole components. In addition, the long process between prototyping and product finalization can be greatly shortened. Nevertheless, the employment of surface-mount techniques in education may be discouraged by both the unappealing part sizes (i.e. handling difficulty) and the excessive cost of commercial supporting equipment. The main objective of this study is to suggest practical and low-cost solutions for all different SMD prototyping/manufacturing stages, which can demystify and render this procedure welcome and easily applicable in laboratory classes.**

Keywords—Surface mount; through hole; PCB; prototyping; laboratory; education

I. Introduction

Surface-mount technology (SMT) is a manufacturing process for electronic circuits where components are directly soldered on the surface of a printed circuit board (PCB), producing the so called surface-mount devices (SMD). Since the first appearance of SMD components for military/space applications back in 1960 [1], it was only in the late 80's where this technology became widespread in commercial products. Since then, it has gradually replaced the classic through-hole technology (THT), where leaded components are inserted into holes in the circuit board, soldered on the opposite side. It is noted though, that both technologies can coexist in the same design, since surface mounting is not suitable for certain components due to their high power rating, required precision or connector/electromechanical type.

The recent prevailing of surface-mount technology can be accredited to certain advantages over its through-hole counterpart. The most important are the smaller component sizes (starting as low as 0.4×0.2 mm) and the ability to place at both PCB sides, leading to higher board density and eventually smaller product forms (Fig. 1). Moreover, while reducing the required number of holes, it is suitable for high-speed fully-automated assembly, which substantially decreases the overall manufacturing cost. It is also a fact that the majority of SMD parts are less expensive (especially passive components) than their through-hole (TH) equivalents (Table 1), so that larger quantities can be sourced at the same cost. Parts are available in cut tape, reel or tray form and can be easily stored even in limited space, preferably ESD protected. It is also noted than many contemporary parts, not only high pin-count ones (e.g. microcontrollers, FPGAs) but also low pin-count (e.g. sensors, power converters) are not available or have limited availability in through-hole form. Finally, SMD components can easily facilitate various state-of-the-art analog circuit blocks for optimum performance (e.g. power conversion, amplification, mixed digital/analog circuitry), which require low trace impedance and suppression of RFI/EMI effects [2]. This is usually achieved by following special layout considerations provided by the manufacturer, which actually enforces actual PCB prototyping (at least in breakout board form (Fig. 1), suitable for breadboard testing).

From the above it is apparent why surface-mount technology has now become the standard for new designs, prototyping and manufacturing. However, its penetration in learning environments (e.g. laboratory classes) is still hampered by two major drawbacks, i.e. small component size which is difficult to physically handle and the excessive cost of commercial supporting equipment for the different prototyping stages. Nevertheless, the recent advent of the Open Source Hardware (OSHW) movement [3] has attracted a substantial number of individuals together with strong support by the electronics-related industry, providing both commercial and custom-made practical techniques and solutions, free resources and documentation. This movement has ultimately rendered SMD prototyping a feasible and appealing alternative to the outdated through-hole practice, which still dominates the educational status quo. In the following sections, these procedures for various prototyping aspects are described and discussed.

II. Prototyping Procedures

The process of SMD prototyping (or small batch production) may follow either of two basic methods, namely hand- or reflow-soldering, depending on the engineer's preference and expertise. These methods will be described in detail in the subsequent sections, following some basic considerations regarding the preceding PCB design.

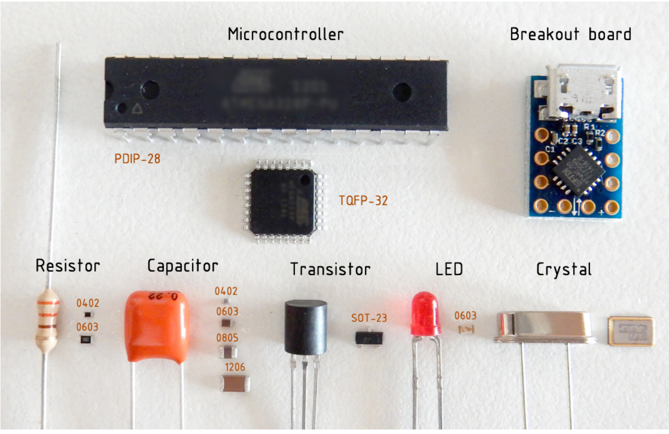

Fig. 1. Surface-mount vs through-hole components and breakout boards

A. PCB design considerations

The transition from THT to SMD design requires some additional considerations regarding the PCB layout. The designer should first check the compliance between the software build-in SMD part libraries and the recommended land pattern (PCB footprint) for each part, usually described in the manufacturer datasheet. When the part is not supported by the library or when there are notable differences between, manual creation or editing is necessary. However, the majority of SMD footprint dimensions are now standardized to their respective package types (i.e. 0603, SOT-23, see Fig. 1), hence the above manual procedure is kept to a minimum. However, the most recent standardization [4] provides three different land protrusion density levels (i.e. 'most', 'nominal', 'least'), affecting land pattern sizes accordingly. As a result, the 'most' density level is recommended for hand-soldered prototypes (larger pads, easily solderable and reworkable), while the other two levels are more appropriate for reflow soldering.

Furthermore, PCB drilling is kept to a minimum in the absence of through-hole parts and it is limited only to necessary vias (for double and multi-layered boards) and alignment holes of connectors/electromechanical parts (e.g. SMD terminals). As a result, in the special case of single layer / SMD-only prototypes, manual PCB manufacturing by conventional laboratory methods (e.g. etching, milling) becomes easier compared to TH-based designs. However, for more elaborate multi-layer designs, commercial manufacturing is advisable, not so because layer alignment and via through-plating is difficult to achieve in the laboratory (need expensive supporting equipment), but mainly because the availability of low-cost manufacturing houses has been significantly increased over the last few years.

B. Manual soldering

The manual soldering method for SMD prototyping is straightforward, requiring a minimum number of inexpensive tools, however, it is mostly demanding in terms of soldering skill and experience. Moreover, the soldering process is usually slow and the quality of the final result is not always guaranteed. The minimum required tools are a set of tweezers for part positioning, a fine-tip, low-wattage soldering iron, a fine diameter solder wire (e.g. 0.20-0.30 mm) and, necessarily, solder flux (e.g. pen) for better workability. Alternatively, hand soldering may be performed by using a hot air gun together with solder paste application, which is preferable for fine-pitch, high-pin-count components (e.g. microcontrollers, FPGAs). It is noted though, that both hand-soldering techniques become considerably difficult to practice on leadless package types (e.g. QFN, DFN), where pads are not accessible by the iron tip, as well as small passive components (e.g. 0603/1.6×0.8 mm or 0403/1.0×0.5 mm), where hot air flow can easily reflow and dislocate nearby components. In any case, a 'third hand' tool that holds components in place while soldering (Fig. 2) [5] and a magnifying loupe for post-soldering inspection are recommended.

TABLE I. SMD VS TH COMPONENT COST[a]

Type	Price per part (€)		
	Description (qty)	***SMD***	***TH***
Passive	Resistor 10K 1% (100)	0.009	0.041
Passive	Capacitor 100nF/≥16V (100)	0.017	0.027
Passive	Inductor 1μH (100)	0.061	0.132
Transistor	2N7002/2N7000[b] (100)	0.061	0.123
MCU	ATmega328P (10)	2.73	2.83
Amplifier	INA114AU (10)	9.45	10.11

a. Lowest price from major supplier, extracted at 28/3/2015

b. SMD: 2N7002, I_d = 120mA, TH: 2N7000, I_d = 200mA

Even if the hand-soldering method is straightforward and inexpensive, it is deemed unsuitable for teaching in laboratory classes, because it strongly depends on personal dexterity rather than providing background knowledge and hands-on training on the present standard manufacturing procedure, i.e. reflow soldering, explained hereinafter.

C. Reflow soldering

The present industry standard for manufacturing the vast majority of electronic devices is the reflow soldering method. This procedure can be divided into three distinct phases, namely solder paste printing, component placement and soldering. For mass manufacturing, all three phases are totally automated with minimal human intervention, however the cost

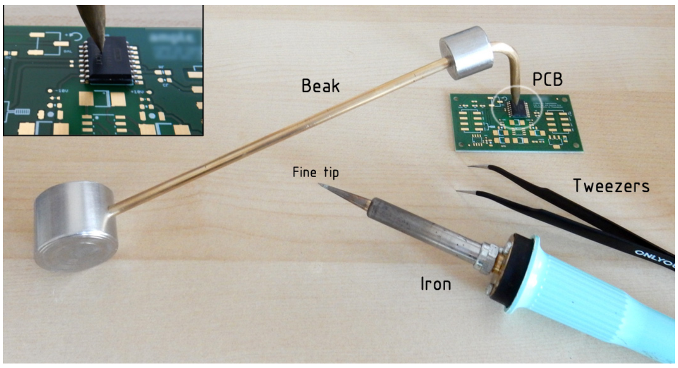

Fig. 2. SMD hand-soldering tools

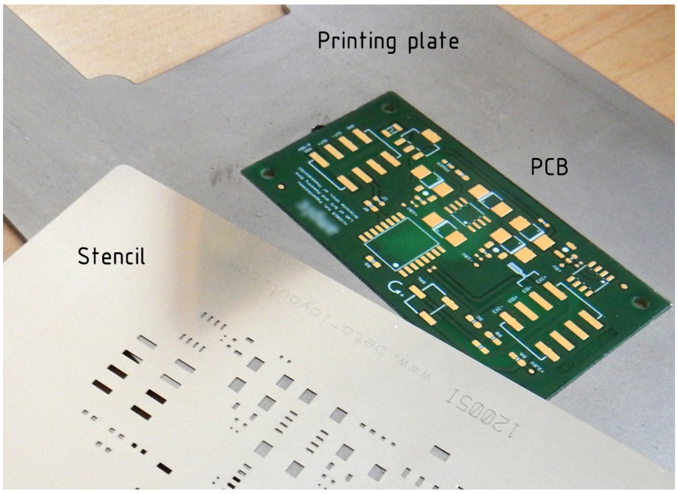

Fig. 3. Manual solder paste printing

of the associated equipment is absolutely prohibitive for educational purposes. For this reason, the following presentation will be focused on low-cost commercial and custom solutions, which can pragmatize this method in any learning environment.

1) Solder paste printing: The first step is to apply solder paste (a mix of powdered metal solder suspended in a thick adhesive medium i.e. flux) on all exposed component pads on the PCB. Solder paste is available in leaded and lead-free compositions, and the contained flux can be rosin-based, water-soluble or ‘no-clean’. The latter has the advantage of leaving nearly no residues after soldering, hence no post-soldering cleaning (e.g. brushing, ultrasonic) is necessary. Moreover, powder particle size is varied (types 1-8), with the smaller ones (e.g. type 3) being more expensive yet appropriate for soldering finer pitch components. It is a fact that solder paste is an expensive substance (about €20 for 50 grams) with a rather short shelf life (about 6 months, if refrigerated), nevertheless the actual amount used each time is negligible. Solder paste may be applied on PCB pads either manually, pad-by-pad, using a syringe or a semi-automated dispenser operated by a foot-switch (starting from €60), or, normally, by covering the PCB with a perforated plastic or metal sheet (stencil) with holes on pad positions (about 10% smaller than respective pad dimension) and then applying paste using a plain spatula (squeegee). The stencil may be ordered together with the PCB from the manufacturing house at an extra or no additional cost. For mounting and precise stencil alignment on the PCB, commercial stencil-printer solutions are available, yet at significant cost (starting from €400). However, low or no-cost custom solutions can be applied instead, e.g. PCB mounting by placing spare PCBs around its perimeter and securing the stencil with tape or more elaborate ones, e.g. a steel plate of the same PCB thickness (normally 1.5 mm) having a laser-cut hole where the PCB is inserted (Fig. 3). The stencil can be then aligned and secured using a pair of neodymium magnets.

2) Component placement: The second phase of the reflow process is the component placement on the PCB pads that have been previously covered with solder paste. Many solutions are herein available, starting from the inexpensive pair of tweezers, which is however the most laborious option, since it requires exceptional hand stability. Manual ot semi-automatic vacuum pens are also available, however they do not provide any serious improvement to the previous practice. At this point, only manual pick-and-place devices can make a difference; they are based on a simple mechanical x-y plane sliding concept, operated by hand, which provides far better stability for accurate positioning. The picking/releasing device is practically a vacuum pen, constrained perpendicularly to the above plane. However, commercial manual pick-and-place solutions are naturally overpriced, starting from €500 up to over €10,000, which may not fit to the educational budget. Nevertheless, since their working concept is trivial, it is relatively easy to resort to custom-built low-cost solutions that can work exceptionally well (Fig. 4, [6]). It is also noted that fully automated benchtop pick-and-place machines have also appeared lately in the market, starting as low as €3000, however, their quality and performance is dubious. Finally, custom-made open-source automated devices have also been developed by outstanding individuals, however they require substantial material and human resources to be replicated and applied in practice.

3) Reflow soldering: Following the component placement, the final prototyping stage is the the actual reflow soldering process. This is performed using a benchtop reflow oven, which should accurately follow a specific temperature profile, compatible to the soldering paste type utilized. This profile should not exceed neither the maximum temperature specification of the solder paste or that corresponding to the most sensitive part (about 250˚C), nor the maximum liquidous phase time of the paste (usually 30-60 seconds, i.e. > 217˚C for lead-free paste). If the above requirements are met, robust solder joints are formed on all component pads and the procedure is successfully completed. Otherwise, the input

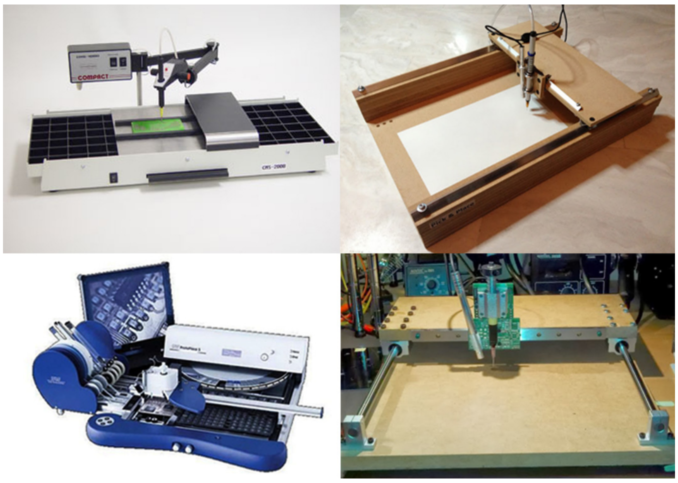

Fig. 4. Commercial (left) and custom (right) manual pick-and-place devices

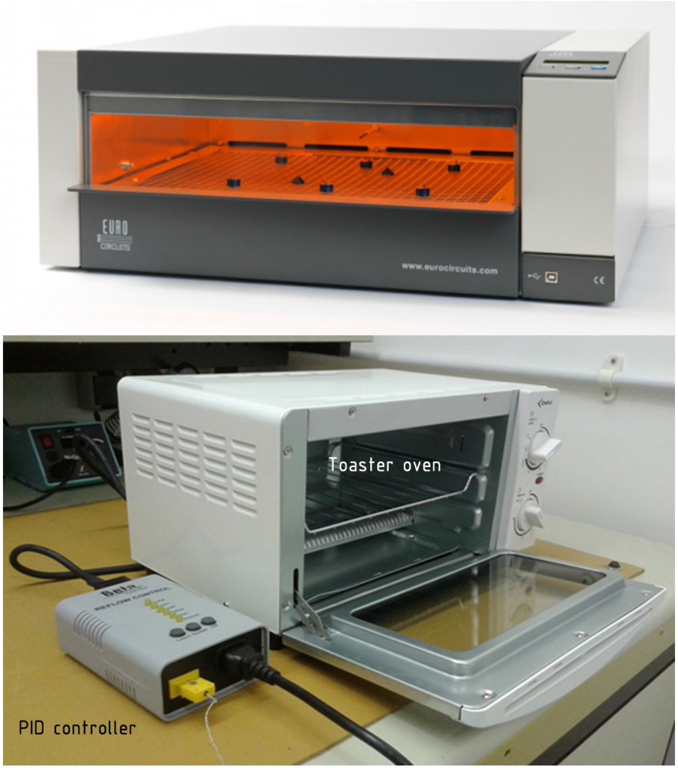

Fig. 5. Commercial (top) and converted reflow ovens (bottom)

profile should be readjusted, especially when adverse effects are observed such as not-reflowed pads, alloy/flux splatters, overheated or tombstoned components. The benchtop reflow oven is hence a vital piece of equipment that should be selected with care. Commercial solutions start from €300, though best quality ovens can cost a few thousand, offering enhanced temperature control, uniformity and repeatability. However, a low-cost alternative is the conversion of a conventional toaster oven to a reflow oven, using a custom or commercial PID temperature controller connected to the oven heater(s) via a solid-state relay, while monitoring the PCB temperature via a thermocouple. This low-budget solution has become increasingly popular, reporting satisfactory results (Fig. 5).

III. Applications

The SMD reflow soldering method has been successfully applied both for prototyping and small batch manufacturing in various civil / earthquake engineering research projects. Fig. 6 shows two prototype examples: (a) a miniature precision strain gauge amplifier for measuring strain in reinforced concrete structural elements (Lab of R/C and Masonry Structures, Civil Engineering Dept., AUTh) and (b) an autonomous MEMS-based triaxial accelerograph [7, 8] for automatic detection and logging of strong motion events (Earthquake Planning and Protection Organization, Greece).

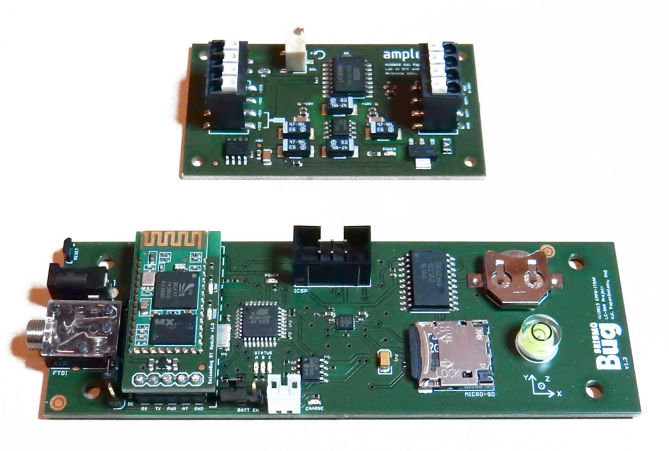

Fig. 6. Miniature strain gauge amplifier (top), autonomous triaxial accelerograph (bottom).

IV. Final Remarks

The numerous advantages of SMD prototyping suggest that the reflow soldering method could gradually replace the outdated through-hole practice in education. This should not only provide broader possibilities for state-of-the-art projects by accessing the full range of the latest component technology, but also in-depth knowledge along with hands-on training on the present industry standard manufacturing procedure. From the present study, it is concluded that the above transition is technically feasible and financially manageable.

References

[1] Himpe, V., “Mastering Surface Mount Technology”, Elektor International Media BV, UK, 2012.

[2] Carter, B. “Circuit Board Layout Techniques”, Texas Instruments, Literature Number SLOD006A, 2008.

[3] Grady, Emcee. "OSHW Definition V1.0". SparkFun Electronics, 2011.

[4] IPC “Generic Requirements for Surface Mount Design and Land Pattern Standard IPC-7351”, Association Connecting Electronic Industries, 2005.

[5] The SMD Beak vpapanik.blogspot.gr/2015/02/the-smd-beak.html

[6] Low budget manual pick and place vpapanik.blogspot.gr/2012/11/low-budget-manual-pick-place.html

[7] Papanikolaou V.K. and Karakostas C.Z. “A low cost MEMS-based accelerograph”, Proceedings of Experimental Vibration Analysis for Civil Engineering Structures (EVACES’13), Ouro Preto, Brazil, 28-30 October, pp. 116-123, 2013.

[8] Papanikolaou V.K. and Karakostas C.Z. “Explicit shakemaps via low-cost instrumentation : The case of Lefkas city”, Second European Conference on Earthquake Engineering and Seismology, Istanbul, 2014.