Using Surface-Mount Components in an Embedded Systems Lab

Erik Brunvand
School of Computing, University of Utah
elb@cs.utah.edu

Abstract—Embedded systems classes and labs can often benefit from having students design their own systems including printed circuit boards (PCBs). These boards can be the basis of either complete small microcontroller systems or add-on boards to existing platforms. However, modern circuit components are very often available only in tiny surface-mount technology (SMT) packages intended for automatic assembly and reflow soldering. This paper describes how we enhanced an embedded systems lab in a cost-effective way to enable students to develop, assemble, and solder custom PCBs that contain SMT components. This allows an enriched hands-on experience, and an expansion of the scope and complexity of the student projects.

I. INTRODUCTION

Embedded systems courses often include a hands-on component where students work directly with embedded system hardware. This may be with existing development boards, or it may include having the students design their own printed circuit boards (PCBs) either as complete, small microcontroller systems, or as add-on boards for existing platforms. In the past, students may have prototyped systems using integrated circuits packaged in dual in-line (DIP) packages, through-hole components such as resistors and capacitors, and with breadboards, wire-wrap, or custom PCBs as the prototyping substrate. These days the components that both professionals and students are more likely to encounter are tiny surface-mount technology (SMT) packages intended primarily for automatic (robotic) assembly and reflow soldering. In fact, modern components such as microcontrollers and other complex integrated circuits are unlikely to be available in anything but an SMT package.

These SMT components are much smaller than through-hole components and result in higher density on a circuit board, but are essentially impossible to use without designing a custom PCB, and are delicate and cumbersome to solder by hand. A typical old-school DIP package has leads on 2.54mm (0.1 inch) centers that are designed to go through holes on a PCB. These, and the through-hole versions of passive parts such as resistors and capacitors, are easily used in any of the previously mentioned prototyping substrates. A typical SMT package, on the other hand, is designed to sit on the surface of a PCB, not to have leads go through holes in the board, and the leads are more likely to be on 0.8mm to 0.5mm centers. Passive devices are also designed to sit on the PCB’s surface and come in standard sizes that range from roughly the size of a grain of rice to something close to a fleck of pepper. Figure 1 is an example of such a board, designed, assembled, and soldered by undergraduate students in a senior project class at the University of Utah.

SMT components are designed to be soldered to the PCB using reflow soldering [2]. This is a technique where a viscous solder paste consisting of a mix of solder and flux is squeegeed onto the board through a stencil. The components are assembled onto the board by nestling them into the small puddles of solder. The board, thus assembled, is put into a reflow oven where the temperature is carefully raised and lowered according to a schedule that matches the characteristics of the solder paste (see Figure 2 for an example reflow schedule). During the reflow phase of the temperature profile, the solder/flux mixture melts and thus adheres the components to the board.

This is a very efficient way of assembling small SMT components on a custom PCB, but one that is not easily supported in an academic lab without some specialized equipment. In this paper I will describe how we enhanced a student lab at the University of Utah to support SMT assembly and reflow soldering of student-designed PCBs in a cost-effective way. This has greatly expanded the scope of projects that students can attempt during a semester. I will include a list of the specific equipment we used. Certainly there are many other choices, but this will serve as just one example. This will also become quickly out of date, but I hope that providing specific
examples and current prices will ground the information somewhat. I will also describe the procedure we use with that equipment that has achieved good results with student projects. My hope is that the description will be detailed enough so that others can use this as a model for enhancing their own labs, and enable a new generation of students to include hands-on design, prototyping, and assembly of custom electronics back into their curricula.

II. EXAMPLE COURSES OVERVIEW

At the University of Utah there are (at least) two courses that deal directly with hands-on embedded systems hardware: CS/ECE 5780 Embedded Systems and CS/ECE 4710 Computer Engineering Senior Project. The embedded systems course is typically taken in the spring semester of the junior year in our Computer Engineering program [3]. The catalog description for this course is:

This class is focused on the principles and practices of modern embedded systems design. In class, we will focus on computer architecture beyond the CPU, fundamentals of the hardware/software interface, techniques for sensing and controlling the physical world, and a few other topics. In lab, we will focus on the MSP430, ARM Cortex-M3, Microsemi FPGAs, and other supporting hardware, to learn how to design, build, and program embedded systems. Labs during the first half of the course will focus on essential topics. The second half of the course will focus on the design and implementation of non-trivial, open-ended projects involving both hardware and software.

Until recently, this class used primarily pre-built FPGA boards (Actel SmartFusion and Xilinx) and pre-assembled add-on boards. The addition of SMT capabilities in the lab made a distinct shift in the type and complexity of the projects. As currently structured, the very first lab in the class is to begin the design a small 1x2” PCB around an MSP430 microcontroller. The students source parts, typically from Mouser [4] or DigiKey [5], and then assemble and solder their boards. These boards are then used in later portions of the course.

The other course where SMT capabilities have had a huge impact is our CS/ECE4710 Computer Engineering Senior Project course. This is the capstone team project course for our Computer Engineering program taken by senior undergraduates (the number for the course, strangely a lower number than the junior-level Embedded Systems course, is a historical oddity). The students in this course will have taken a semester-long senior project planning course in the spring, and the teams from that course continue in the Fall semester to implement a student-chosen project. As senior projects in the Computer Engineering degree program, senior projects are required to have a mix of both hardware and software designed and implemented by the students. Again, until recently the hardware components of the senior projects were largely small extensions of existing embedded system platforms. With the addition of PCB design skills and SMT assembly and soldering to the lab, six out of seven teams in the most recent (Fall 2014) offering of the course included a student-designed PCB that included SMT parts as part of their project. Some examples of student-designed boards from this class are seen in Figures 1, 3, and 4. Figure 5 shows a student working in the lab during Fall 2014. While it is difficult to quantify and compare the quality and complexity of student projects from year to year, visitors to the demo day at the end of Fall 2014 were greatly impressed with the projects from this year, many of which would have simply not been possible without the ability to work with SMT components on a custom PCB.

III. LAB ENHANCEMENTS

The first requirement for working with custom PCBs and SMT components is to introduce PCB design tools to the students. These tools typically start with a schematic view of
Fig. 4. An assembled board with debugging wires attached. This board was designed, assembled, and soldered by Brent Mellor and Chase Wilson. The processor chip is a Cypress PSoC 4 “programmable system on chip” in a TQFP 44 package with 0.8mm pin pitch. The chip above the processor is a CAN-bus controller chip in a 20-pin TSSOP package with a 0.65mm pin pitch. The board was designed using Altium Designer. [6]

Fig. 5. A student working at the soldering station.

a circuit, and then allow the designer to produce a layout of a PCB that implements that schematic. This may involve having the designer draw new “footprints” that define how the SMT components will sit on the board to be included in the PCB layout. We have used three different PCB tools in different contexts at the University of Utah, and there are many others out there to choose from.

- **Eagle CAD**: Eagle is widely used by hobbyists and is supported by hobbyist web sites such as Sparkfun and Adafruit who publish Eagle footprint libraries for most parts that they sell. Eagle has a freeware version for non-profit use that restricts designs to two signal layers and a 4x3.2 inch board area. This may be completely sufficient for student lab needs. All but one of the boards designed in my Fall 2014 Senior Project class would have met these specifications. A multi-user license for an accredited educational institution has costs that depend on the number of licenses and features desired, but as an example Eagle standard (99 schematic sheets, 6 signal layers, 6.4x4 inch board area) for 10 users would cost $1,230 as of April 2015. [7]

- **KiCad**: This is an open-source community-supported tool that runs on a wide variety of platforms. At least one of my student teams used KiCad because of the open-source nature, but the support, especially in terms of footprints for non-common parts, is less than with Eagle. [1]

- **Altium Designer**: This is a professional-grade PCB tool designed for large high-volume professional users. It is considerably more expensive than Eagle, but is widely respected as one of the best professional tools. Although there is no specific university program, universities may be able to negotiate an educational discount as we have at the University of Utah. Footprint support is limited, although the “footprint wizard” is very efficient, and there are third-party companies that provide device footprints at a cost.

Even with the ability to make SMT-capable PCBs, some restrictions on what types of SMT components are used can increase the success rate. For example, in two-terminal components (resistors, capacitors, LED’s, etc.) we recommend using 1206, 0805, and if necessary, 0603 components. Smaller components are certainly possible, but dramatically increase the soldering difficulty (at the extreme, a 01005 component is barely visible at 0.4 mm x 0.2 mm (0.0157 in x 0.0079 in)). In three-terminal packages, an SOT-23 is a good usable size if possible. For larger components, we recommend that students to not go below a 0.5mm lead pitch, and to not mess with ball grid array packages that have a set of issues all their own, especially for rework. Packages that have been used with good success include SOIC, SSOP, TSSOP, QFP, LQFP, and TQFP. A good overview of SMT packages can be found on Wikipedia [8].

Once the PCBs are designed, they must be fabricated. Partly because of the small SMT parts, the circuit traces are typically too fine for easy fabrication in-house. Also, with denser designs using many small components, at least a two-layer board is usually required. Fortunately, there are many cost-effective ways to get high-quality small-volume PCBs fabricated. These typically have minimum trace widths/spacing of 6-8 mils in 1oz copper. Some fabrication houses that my students have used include:

- **OSH Park**: This is a “panel sharing” site where user designs are aggregated until a full PCB panel is ready to fab. This amortizes the setup costs for PCB fabrication among the users and keeps costs low. The cost is $5.00/in$^{2}$ for a two-layer board and $10.00/in$^{2}$ for a four-layer board with 6mil minimum width traces and three copies of the board are included in that price. Turnaround is around 12-14 days. OSH Park takes Eagle files directly, or Gerber files from other PCB design tools. [9]

- **CircuitGraphics**: This fabrication facility also has a panel sharing option at $3.25/in$^{2}$ per copy of the board.
Circuit Graphics has an 8mil trace minimum and a five-day turnaround. [10]

- **Advanced Circuits**: This fabrication facility has an option for a single larger two-layer board of up to 60in² for $33 (four-layer boards are $66). Boards can have 6mil traces, and there is a five-day turnaround. [11].

### A. Equipment List

A photo of the side of the student lab devoted to SMT assembly and soldering is shown in Figure 6. The following is the list of equipment that we installed in our lab to support assembly and soldering of SMT components on student-designed PCBs:

- **Stencil definition**: In order to apply the solder paste only to the pads on the board where components will be soldered, a solder mask needs to be cut. This is a stencil that has holes only where the solder paste should be applied.

- **Stencil cutting**: To actually cut the mylar for the stencil, we use a hobbyist vinyl cutter that is geared to scrapbooking and other crafts. Specifically we have installed a Silhouette CAMEO cutting machine [13]. This consumer-grade cutter costs around $270 and easily cuts the 4mil mylar that we use for stencils [14] (see Figure 7).

- **Assembly**: To assemble the components onto the board (after the application of solder paste), it is very helpful to have an inspection microscope. This is a relatively low-powered binocular microscope with a ring light around the lens. We find that it is important to have a long working distance to the lens and a boom stand to make it easier to move the scope over the board. There are many microscopes that would work - we chose a “Circuit Zoom Stereo Microscope” from Amscope [15]. Prices depend on features - our scope was approximately $570.

- **Soldering Oven**: The reflow soldering action takes place in a reflow oven. Reflow soldering can actually be done on a hot plate or even a frying pan (there are many hobbyist web sites that will describe how to do this). However, a reflow oven works much more reliably and can execute a specific reflow temperature profile like that in Figure 2. Large commercial high-volume reflow ovens can be extremely expensive, but we have found that small inexpensive bench-top “mini reflow ovens” work well for our lab. We have had good success with small ovens from SMTmax that cost from $550-$900 depending on size [16].

- **Rework**: Even with the most careful schematic design, first-spin PCBs often have issues, and components can also sometimes be damaged in the assembly process or over time and need to be replaced. It’s also possible that for some parts of a design hand-soldering is needed for some reason. Because of this, good soldering stations are desirable both for hand-soldering and for rework. We have found that two types of soldering stations are particularly useful.

  - **Soldering tweezers**: Especially for two-terminal components such as SMT resistors, capacitors, and LEDs we have found that soldering tweezers (hot tweezers) are extremely useful. This is essentially a tweezer with a soldering iron in each arm of the tweezer. With a fine tip, it’s easy to grab the component by the ends with the hot tweezers and remove it from the board when the solder melts, or to place the component by grasping with the hot tweezers and placing into the solder puddles already on the board. There are many hot tweezers to choose from, and nearly all high-end soldering stations have a hot tweezer option. We have found that the inexpensive tweezers from Circuit Specialists work well at only $90 [17].

  - **Hot air soldering**: For point-specific reflow, it is possible to use a hot-air soldering system. This is essentially a high-temperature small-tipped heat gun. Using a hot-air system one can quickly loosen the solder on a component that needs replacing, or reflow the solder to attach a new component. We have used a station from X-tronic that includes both a hot-air system and a fine-pitch traditional soldering iron and starts at around $150 [18].

- **Cleaning**: After reflow, there is often flux residue on the boards. When using solder paste with water-washable flux, we have found that a commercial steam cleaner makes a very effective cleaning system for the circuit boards. For example, a hand-held steamer from DBTech costs around $35 [19].

- **Other supplies**: Other consumable supplies that are needed for SMT soldering include solder paste (we like the water washable type [20]), flux pen for hand-soldering and rework [21], blank mylar for stencils [14], tweezers for assembling components onto the board [22], blank circuit boards for holding the board that is to have solder paste applied, a putty knife for applying the solder paste through the stencil, and a small refrigerator for keeping the solder paste cold during storage.

### IV. Example and Procedure

The following is an example of the procedure for assembling and reflow-soldering a small circuit board. This example uses the board shown in Figure 1 that was designed and assembled by undergraduate students Dan Willoughby and Zach Toolson. The procedure starts with a designed and fabricated PCB. In this case Dan and Zach used KiCad, but any PCB tool will work if it produces Gerber output files. Once the board
has been fabricated and the components been acquired, the procedure is as follows:

1) Convert the Gerber solder mask layer in your PCB description to a “graphtec” format using gerber2graphitec [12].

2) Cut a stencil of the converted solder mask on 4mil mylar using an automated cutter such as the Silhouette CAMEO cutter (Figure 7).

3) Prepare the board for the application of solder paste by taping it down nestled between blank circuit boards. This makes an even surface that extends past the board to make the squeegee process easier. This can be seen in Figure 8. The purple circuit board is surrounded on all sides by yellow blank circuit board material to make a smooth surface for the solder paste squeegee.

4) Carefully place the stencil onto the board, lining up the openings over the pads on the board. Tape this down too. The inspection scope is very useful in this step to make sure that the openings in the stencil are properly aligned with the solder pads on the board.

5) Apply the solder paste near one edge of the board, ready to be deposited on the PCB through the stencil with the squeegee. See Figure 8 again.

6) Use a plastic putty knife to squeegee the solder paste across the stencil to apply the paste to the board. This is similar to screen printing if that helps describe the step. We have found that it is best to apply the solder in one steady pass rather than try to move back and forth across the stencil (see Figure 9).

7) Carefully lift the stencil leaving the solder only on the solder pads of the PCB.

8) Verify with the scope that each copper pad actually has
solder on it. You may need to place a small amount of solder paste on some pads using a toothpick or tweezers.

9) You may also need to remove extra solder paste that is bridging multiple pads. You don’t actually have to be perfect here, especially for fine-pitch pads. The reflow process will tend to draw the solder to the pads and away from the coated parts of the board. But, if there are obvious blobs, it’s helpful to remove them with a toothpick or very fine tweezers.

10) Clean the stencil and putty knife with the steam cleaner so that they’ll be ready for the next use.

11) Now use tweezers to carefully place the components onto the board, nestling them into the solder paste (Figure 10). The paste is sticky and will hold the components into position. Components should be placed as carefully as possible, but it’s all right if they’re slightly crooked. When the solder flows during the reflow process it will tend to pull the components into place through surface tension. The inspection scope is very useful in this step, especially for components with a large number of pins. It’s also helpful to have a copy of the PCB layout handy to remind you where each component goes, and to check off components as you go.

12) Once the components are placed, carefully transfer the board to the reflow oven. The basic reflow schedule is likely quite effective, but you may want to adjust the schedule to match the characteristics of the solder paste you are using. Figure 11 shows this board on the oven tray ready to be slid into the oven.

13) Fire up the oven to go through the reflow process. If your oven is not vented to the outside (ours is not), place a filter fan such as you would use for soldering behind the oven to draw the fumes through the filter.

14) Allow the oven to cool completely before opening. This will allow the components and the solder to set properly. In our oven, this means letting the oven cool to at least 70°C before opening the oven to remove the board. That’s still pretty toasty, so you can wait longer if you like.

15) Inspect the resulting soldered board using the scope to see if there are any obvious issues before applying power. If there are issues, use the hot-air or hot tweezers to rework and fix the issues.

16) Clean the board using the steam cleaner.

17) Apply power and enjoy your completed, working board. In Figure 12 the LEDs indicate at least basic functionality of this particular board.

V. Conclusions

Prototyping with modern integrated circuits and circuit components essentially requires the ability to deal with SMT
components. Many current parts are, in fact, only available in SMT-type packages. Fortunately, using these parts with a reflow soldering setup is not all that difficult, or expensive. Certainly care must be taken when using parts with fine pitch leads, or that are physically tiny, but our experience is that this is very feasible in an undergraduate embedded systems lab. A production setup for SMT and reflow would be extremely expensive, but a student lab setup such as described here can be installed for under $2000 (not including the optional expense of non-free PCB design software). Even with a few added extras and upgraded equipment from the basic suggestions in this paper, the cost is unlikely to exceed $3000 for the capital equipment.

We have found that the re-introduction of hands-on system prototyping and assembly that is enabled by this SMT and reflow equipment has made a noticeable difference in the quality and complexity of the projects our students pursue. This allows student projects to develop exactly the board/system they need rather than settle for the board they can buy. It also flexes their design skills to imagine the complete system with all the required components to make the board work. Finally, there is a hard-to-quantify sense of ownership and accomplishment that the students get from deploying their own board in their projects that has seemed to really raise the level of excitement in the lab. This seems to be an essential “maker” capability for modern embedded system design and prototyping.

ACKNOWLEDGMENTS

I have benefited greatly from the advice of graduate student Anh Luong who developed a similar SMT/reflow procedure for his research group at the University of Utah. Also, my students in CE/ECE 4710 from Fall 2014, especially Phillip Bradbury, Nathan Hansen, Brent Mellor, Zach Toolson, Dan Willoughby, and Chase Wilson, bravely used this setup in its preliminary form and discovered many interesting tweaks and corrections to our initial flow.

REFERENCES