# USING SURFACE-MOUNT TECHNOLOGY IN STUDENT PROJECTS

**Peter E. Goodman**
*Indiana-Purdue University Fort Wayne, Fort Wayne, Indiana*

## 1. ABSTRACT

This paper describes the techniques which have been adopted and the lessons learned as a result of incorporating surface-mount technology (SMT) in student projects (capstone projects and projects for other courses) at IPFW. Surface-mount technology has become so pervasive in the electronics industry that many of the parts used in student projects are no longer available in any other form. At the same time, surface-mount technology is often thought to be unsuitable for prototyping and for student projects because of the setup costs associated with industrial surface-mount assembly, and because of the perceived difficulty of handling the tiny components. These perceived difficulties need not be barriers. The techniques described here are neither difficult nor expensive.

## 2. INTRODUCTION

At IPFW students are required to take two courses which involve making and assembling PC boards. As sophomores, they take an Electronic System Fabrication course, in which each student is required to design and fabricate two to three PC boards and solder parts to them to produce working (hopefully) projects. Each senior must complete a capstone Senior Design Project course before receiving the BS degree. These projects have traditionally used through-hole technology almost exclusively surface-mount technology is perceived as much more difficult and as requiring a large capital investment. The electronics industry, the future employers of our students, has been moving ahead with SMT for over twenty years because of its many advantages, and this trend has accelerated in recent years. We have managed to avoid SMT while the world moves ahead, but we will not be able to avoid it much longer. SMT is becoming so pervasive in industry that many advanced integrated circuits, such as digital signal processors (DSPs) and microcontrollers, are no longer available as through-hole parts. If advanced parts are to be used in student and faculty projects, there is no longer any alternative to SMT. We must find economical and feasible ways to move ahead. Fortunately, that is easier than it may seem.

## 3. CHEAP SURFACE-MOUNT TECHNIQUES

The preferred method of assembling surface-mount parts to a PC board begins with application of solder paste to the pads (a pad is a small copper area at the end of a trace, to which a package pin is connected by soldering). Each pad receives a small amount of paste, which is a suspension of tiny beads of solder in a viscous liquid flux. The surface-mount parts are then placed on the pads where they are to be soldered, and are temporarily held in place by the solder paste. When all parts have been placed, the board is placed in an infrared or convection-type reflow oven, and heated according to a well-controlled temperature profile. The board is first heated to a temperature which is not high enough to melt the solder beads, but is sufficient to activate the flux. It is held at this temperature for 60-90 seconds while the flux cleans the pads. The temperature then continues to rise to the melting point of the solder beads, and reaches a peak temperature about 30 degrees C above the melting point. It remains above the melting point for 60-90 seconds, during which time the solder beads melt and reflow to form solder joints. The surface tension of the molten solder pulls the parts into good alignment with their pads. The oven is then cooled to the melting point, and then cooled as rapidly as possible without causing excessive thermal stress.

In industry, infrared or convection-type ovens, designed for this application, are used. Small reflow ovens cost upwards of $4000. Larger, more sophisticated ovens are available on the resale market for around $20,000. As a result, some are tempted to hand-solder surface mount parts with a fine-tipped soldering iron, a lighted magnifier or microscope, and a steady hand. This may work if there are no very small parts with very fine pad spacing or large pad counts, but it is a recipe for failure if it is tried with the tiny quad flat pack integrated circuits or 0402 or 0201-size resistors and capacitors. Hot-air soldering stations, which are used for repair of surface-mount assemblies, is also sometimes used. Hot-air soldering equipment is less costly than an industrial or laboratory reflow oven, but it is by no means cheap. Fortunately, there is a better way.

A number of authors (LaCoste, 2004; Rooks, 2006) have described the use of an ordinary toaster oven as a reflow oven. While it appears that most any toaster oven could be used in this way, some are better than others. Convection-type ovens offer the most even heating, which is a definite advantage. One source, Stencils Unlimited, specifically recommends the Euro-Pro TO289, so we purchased one of these for $50 at the local Target store.

All toaster ovens are equipped with a timer and a thermostat, but these are very inaccurate and useless for reflow soldering. Fortunately, there are three ways to obtain temperature control: the inexpensive way, the cheap way, the cheaper way, and the cheapest way. The cheapest way is to use a piece of a "temperature marker stick," which changes color and reflows at the melting temperature of the particular solder paste being used. These are available from Stencils Unlimited. Shave a small piece off of the temperature indicator and place it on an unused area of the PCB. Heat the oven, watching through the door, until the temperature indicator changes color and reflows. The temperature marker helps to make sure that the reflow temperature is reached, but it does not help follow the optimum temperature profile for the solder paste.

Better control is achieved at a slightly greater cost. A type-k thermocouple (such as the B&K Precision TP-3, available from Mouser Electronics (Mouser Electronics, 2006) for $19) may easily be connected to a simple interface amplifier using an Analog Devices AD595 Monolithic Thermocouple Amplifier with Cold Junction Compensation (obtained as a sample), yielding an output of 10 mV / degree C (in other words, 200 degrees C = 2.00 V). The output voltage is monitored using a digital voltmeter, and the oven is manually cycled on and off with stopwatch in hand, in order to follow the temperature profile. The cheap way involves connecting the thermocouple amplifier to a microcontroller development board, which reads the temperature and cycles the oven on and off via a relay to keep it on the temperature profile. This adds less than $100 to the total cost.

The inexpensive way is to simply order a temperature controller, which is also available from Stencils Unlimited. The cost for this is $246. Any of these techniques should give adequate results, but at IPFW a Silicon Labs C8051F020DK-U Development Kit is used, along with a custom interface. The schematic diagram and PCB layout for the interface, which includes the thermocouple amplifier, a relay, and a relay driver, are shown below:

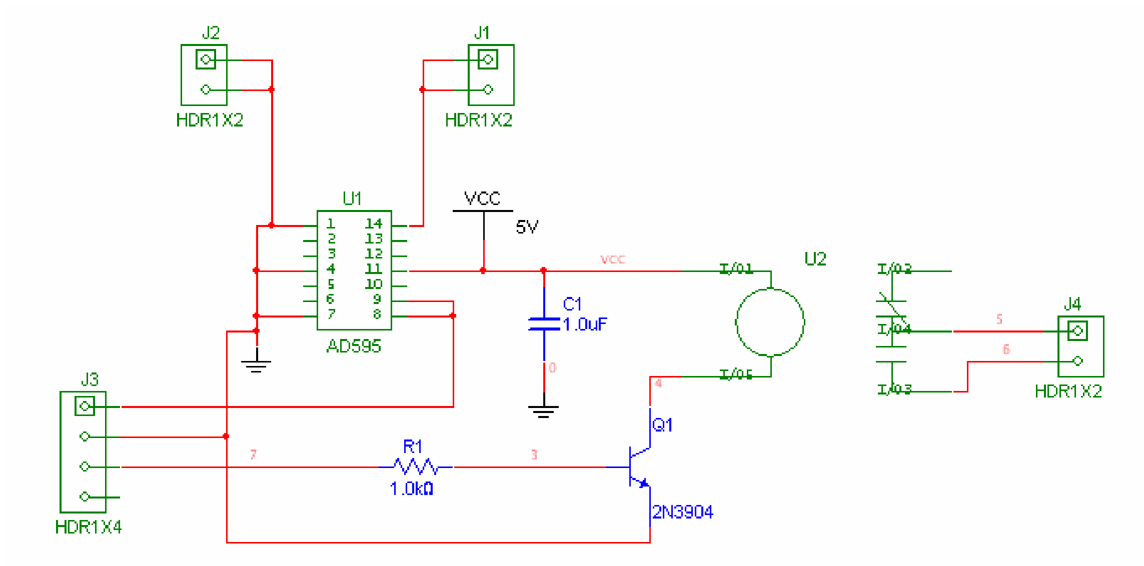


Figure 1: Oven Controller Interface Board

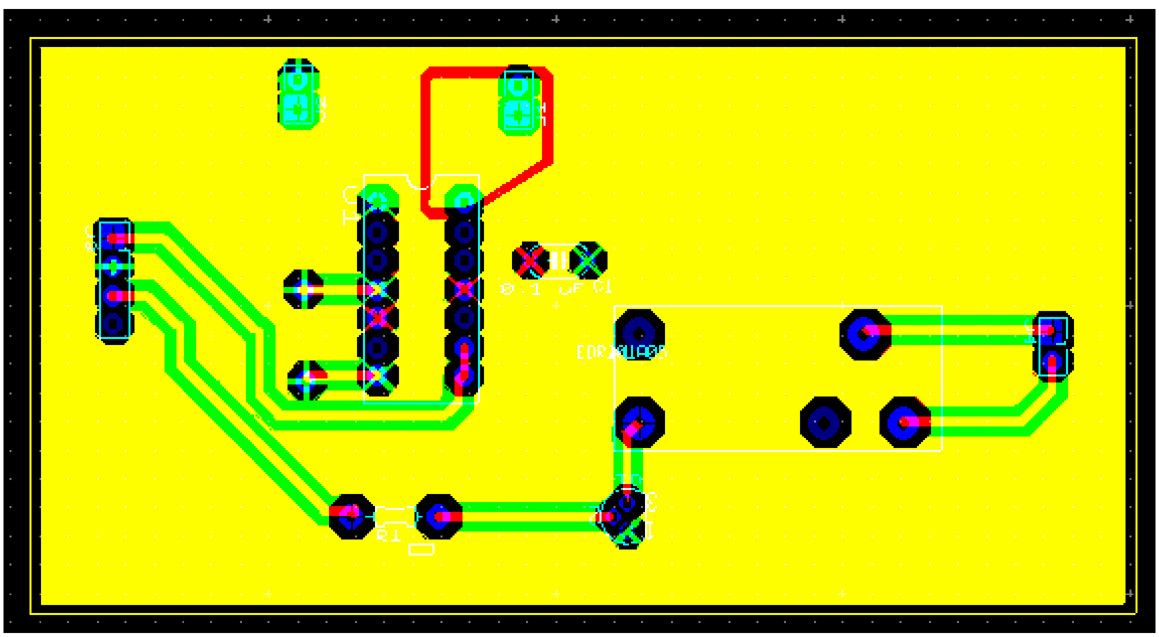

Figure 2: Oven Controller Interface Layout

The most convenient and easiest way to apply solder paste is by silkscreening, using a laser-cut stencil. The stencil is designed as part of the PCB layout, and is made using an expensive laser stencil cutter. Prototype stencils can be ordered from vendors like Stencils Unlimited or PCB Express (PCB Express, 2006) for about $150. Fortunately there is a cheaper way here, too.

Solder paste is available in fine-tipped syringes, and these have been found to work very well for PLCC-68 IC packages, and resistors and capacitors as small as 0201. Paste was applied for PLCC and 0805 packages using a "head loupe" binocular magnifier ($5.00 from the local Harbor Freight Tool store) with no difficulty; for smaller packages a microscope was used. Experience has shown that unsteady hands may be stabilized by simply holding the syringe in both hands. If paste is inadvertently deposited between pads (anywhere else it should not be), it can be removed by scraping gently with an X-Acto knife.

For fine-pitch parts like quad flat-pack IC packages, the solder paste may be applied using a "rework"stencil. This is a small stencil, cut to apply solder paste for a single IC package instead of an entire board. It is called a rework stencil because it is normally used when replacing a defective IC, not for assembling an entire board, but there is no reason rework stencils cannot be used to apply paste to only the difficult parts of a PCB (for example, one or two TQFP footprints) with a syringe used to apply paste everywhere else. Rework stencils are available from Mini Micro Stencil, Inc. (Mini Micro Stencil, 2006) for about $80.

When the solder paste has been deposited, the parts are placed on the board. Stencils Unlimited offers a handheld vacuum pick-and-place tool for under $20, but tweezers seem to work better. Resistors and capacitors are shipped in a recessed tape (for automatic assembly), and are removed from the tape by scoring the plastic film which covers the tape recesses which hold the parts. The film is scored on all four sides of a recess, and carefully lifted off with the tip of the X-Acto knife. The part is then carefully lifted out of the recess with the tip of the knife, and placed on a surface where it can be grasped with tweezers for placement. This is quite easy with 0805-size parts, but slightly more difficult with 0402 size parts. The smaller parts are so small and light that there is a risk of catapulting them across the room while lifting them from the tape. 0201-size parts are quite difficult to handle for this reason, but once they are in the tweezers are only slightly worse than 0402-size. Each part should be gently pressed into the paste, to make sure it sticks, after making sure it is aligned correctly.

Here are several photographs of parts which were reflow soldered in this way:

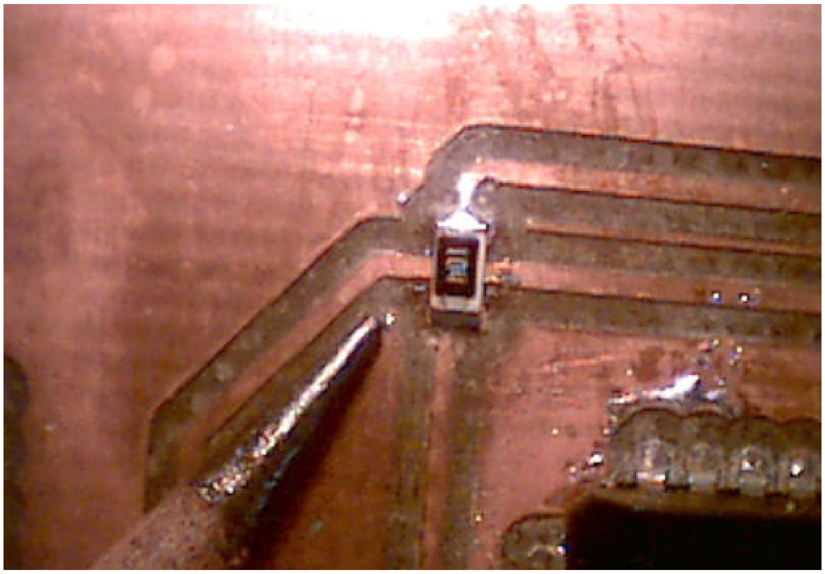

Figure 3
0805 Resistor at 10X Magnification With 10-mil Trace Underneath

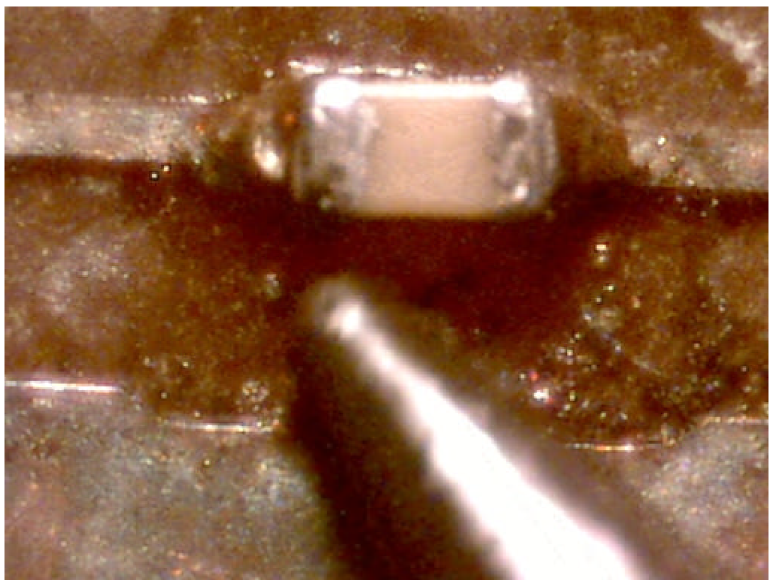

Figure 4: 0402 Capacitor

Figure 5
0201 Capacitor

Of course, before using these cheap assembly techniques, one must have a PC board to assemble. Project PC boards can be made to order from customer-supplied Gerber files by vendors like PCB Express and Stencils Unlimited. The cost is about $150, and includes a solder stencil. Boards made by these sources have solder masks and are silkscreened with component outlines, which makes assembly a little easier and may help justify the cost. It is also possible to make single- or double-sided boards in your own lab using traditional photomasking and chemical etching techniques, but it is messy and disposal of the used chemicals is problematic. At IPFW, we have eliminated chemical etching and now use a CNC milling machine to cut patterns directly into copper. These machines are available from LPKF (LPKF Laser and Electronics, 2006), T-Tech (T-Tech Inc., 2006), and other sources, but all are similar in many ways. We are fortunate enough to have a T-Tech Quickcircuit 5000, which works very well. The cost including all accessories (such as a dust collection system, which is called an “accessory” but is really a requirement) was about $10,000.

The CNC mill was by far the most expensive item in our PCB lab, but it is much more convenient to use than a chemical etch process, and more economical than having boards made by outside vendors. The documentation provided by T-Tech is somewhat lacking, and there are a few “tricks” to using it effectively. Many of these, and detailed setup instructions, are available on the author’s website. Here are some of the more important ones:

First, before ever milling a board, we apply a tin coating to the board. We use a product called Tinnit, which is made by Datak and available from a number of sources (Electronics Express, for example) and is very easy to use. This is currently the only wet process in our lab, a slight inconvenience which is easily outweighed by the marked improvement in solderability.

Setting the depth of the cutting tool, that is, how deeply it cuts into the copper layer and substrate, is critical for good results. The best way to do this so far is to press the head down manually with the spindle off, until it is all the way down and the “shoe” is pressed against the board. The tip of the tool should not touch the board, having been adjusted to a negative depth before lowering the head. The height adjustment (a "knurled nut" on the T-Tech machine) is then used to slowly bring the tool down, while watching through a head loupe or magnifier, until the tip of the tool just touches copper. The machine is now set to a depth of zero. On the T-Tech machine, each click of the height adjustment lowers the tool (increases the depth) by 0.4 mil, and half-ounce copper has a thickness of 0.7 mil, so three or four additional clicks (four or five for one-ounce copper) is just about right if the tool is an endmill. Increasing the depth further degrades quality and tool life.

If the tool in use is a "pointed" tool, such as the T1 mill sold by T-Tech, the width of the cut depends on the depth. The T1 tool has a 60 degree point, so if it had a perfect point the width of the cut (at the copper surface) would be equal to the depth. Thus, if the procedure described above were followed to set the depth to zero, thirty additional clicks would result in a 12-mil wide cut. Experience, however, shows that around 16 clicks yields a 10-12 mil cut. The discrepancy seems to be due to the fact that the point of the tool is not perfectly sharp, it is somewhat blunt. If compared to an perfectly sharp tool with a 60 degree point, the real-life tool is around 5-6 mil shorter, as if one had ground that amount off the point of a perfect tool. That means that the width and depth of the cut are not equal, it is about 5-6 mil wider than it is deep.

There are several ways to check the width of the cut. The easiest and least reliable way is to use feeler gauges, which are available from T-tech. The most accurate and the most expensive way would probably be to use a microscope with a measuring scale reticle. There is a third way, which costs nothing and seems to be pretty reliable: First, after adjusting the depth, start the machine and make a ½ inch (or so) horizontal cut under manual control (the T-Tech machine has a "jog" function, available from a pulldown menu for this sort of thing). Raise the spindle, but do not turn it off. On the "jog" dialog, select "incremental" and set the increment equal to the desired width of the cut. Click the up arrow twice, to move the spindle twice the desired width, then lower the spindle and make a second cut parallel to the first. Raise the spindle, stop the spindle, and move it away so the cuts can be inspected with a head loupe or magnifier. You should see two cuts of equal width, with a strip of copper between. If the width of the copper strip is equal to the width of the cuts, you have adjusted the depth (and therefore the width) correctly. If the copper strip is wider than the cuts, the cuts are too narrow and the depth must be increased. If the strip is narrower than the cuts, decrease the depth and test again.

When milling (that is, when removing copper to isolate traces as opposed to drilling or routing holes), the spindle should run at its maximum speed. The speed at which the spindle moves across the board when milling should be slower than the manufacturer's recommendation, especially for very fine cuts. 2 – 5 inches per minute seems to work well, and does not increase the time required to mill a board that much. These practices do not degrade tool life, either. Tool (that is, the bit, not the machine) life is generally a fixed number of inches regardless of spindle RPM, because the amount of copper which can be removed by one revolution is fixed. Running the spindle at high RPM but cutting slowly improves quality, but does not wear tools out faster.

Tools are, of course, available from the manufacturers of milling machines, but other sources such as Think and Tinker (Think and Tinker, 2006), which was found while reading posts on the Yahoo "Homebrew PCBs" group (Homebrew_PCBs, 2006), may offer comparable tools at lower cost.

## 4. AN SMT PROJECT

The surface-mount techniques described here are being used this semester in the Electronic Fabrication course, ECET-296, which is the capstone course for students earning the A.S.E.E.T degree. One of the projects which students are required to build (and to design their own PC boards) is a software-defined radio receiver operating in the "20 meter" (14 MHz) Amateur Radio band. It is based on a popular Amateur design, the "SoftRock 40", version 5 (The American QRP Club, 2006), modified to use surface-mount parts throughout (with just a couple of exceptions). SOIC-14, SOT-23, and 0805 packages are used, so it should not be too difficult for the students to build, but it is not trivial. The cost of parts is about $18.00 per radio. The schematic diagram and PCB layout are shown below:

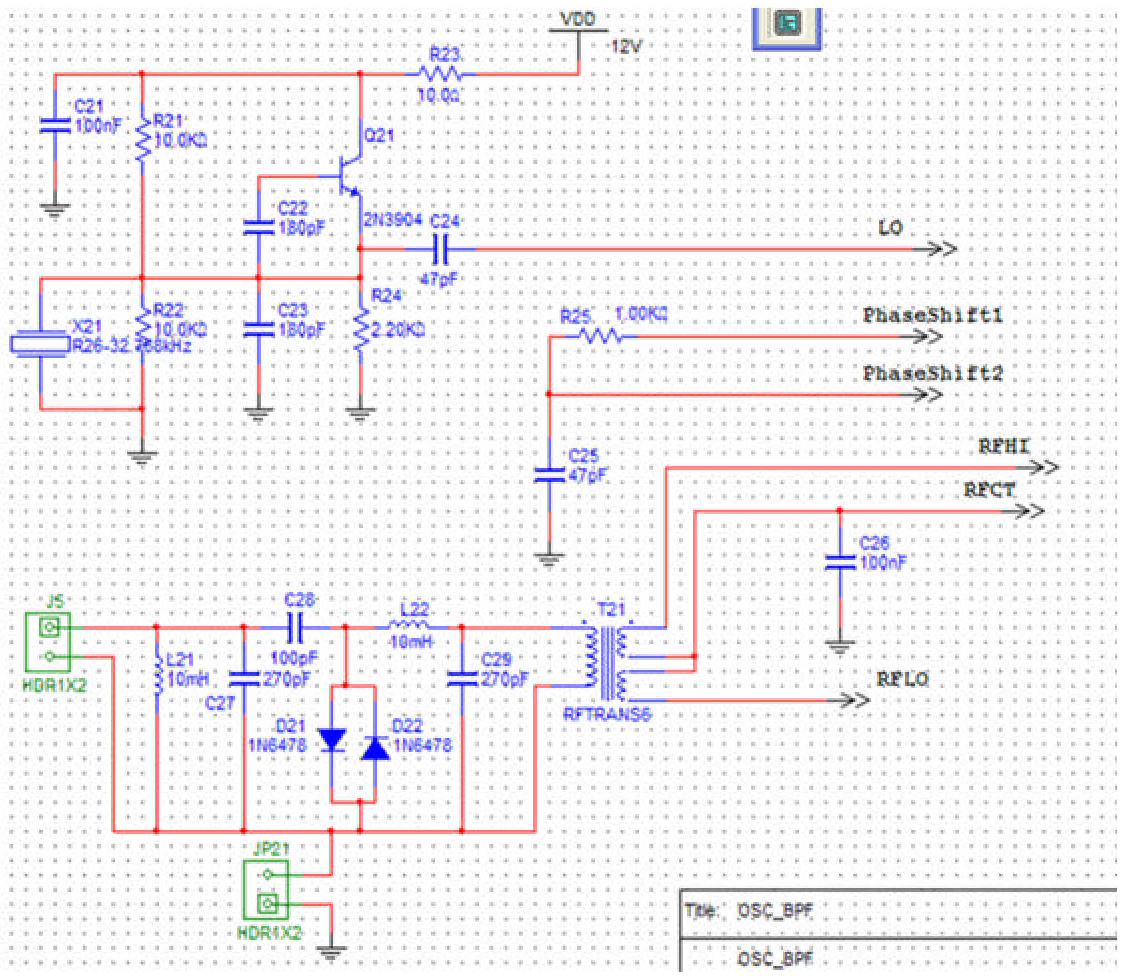


Figure 6: SDR Project, Page 1

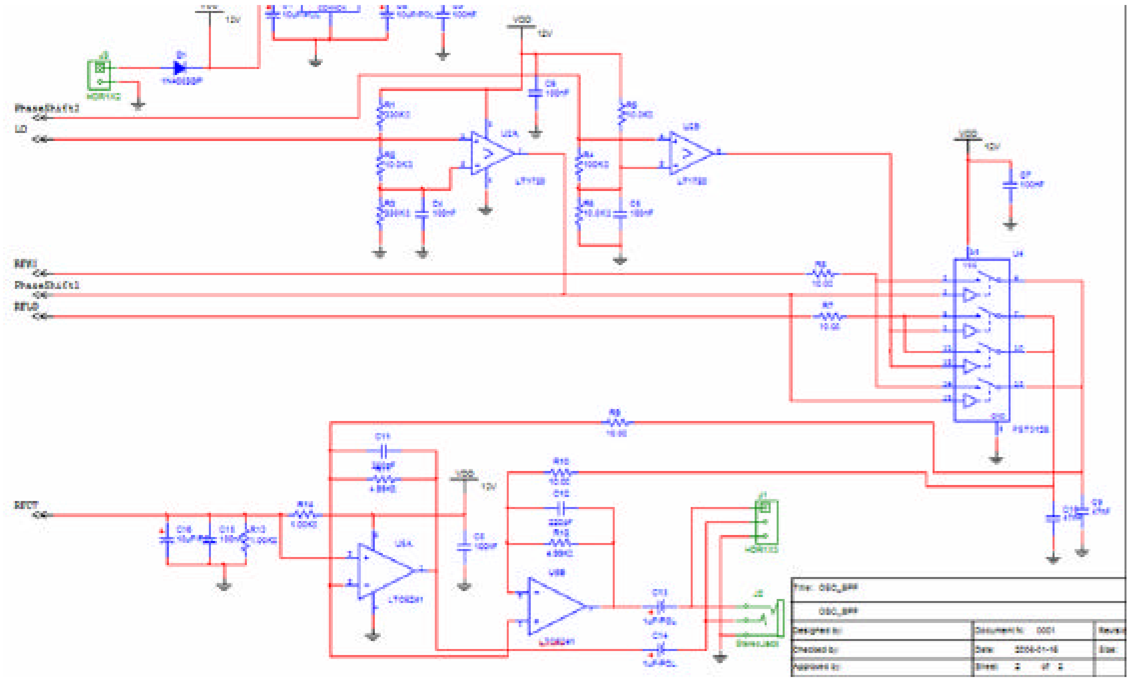


Figure 7: SDR Project, Page 2

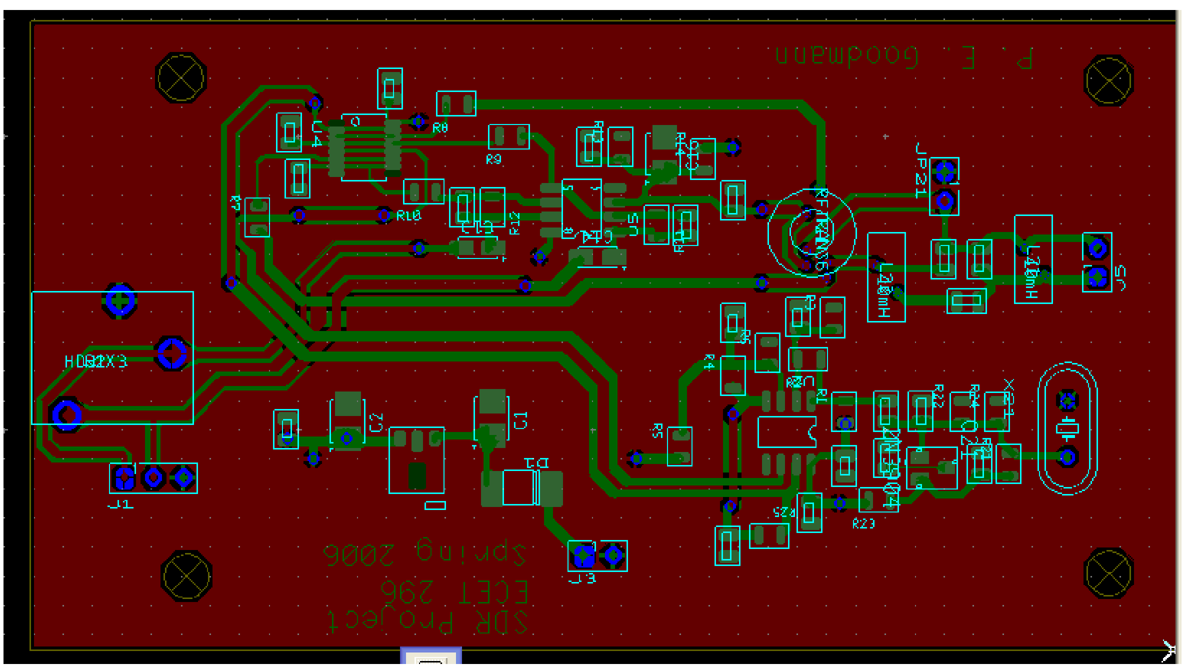


Figure 8: SDR Project Layout

## 5. CONCLUSION

Although the traditional path which led technically-inclined teenagers into careers in electronic engineering and technology, amateur radio and tinkering with electronics during the high school years, followed by a university education in engineering or technology, has nearly disappeared, it can be restored. Students must be exposed to the "fun, cool" side of electronics by involving them in construction projects as early as possible. Amateur radio projects and demonstrations can be an important and useful way to do this. The techniques used in student projects must be brought up-to-date by incorporating surface mount technology, but in ways which are economical. Fortunately, these techniques are available.

## REFERENCES


LaCoste, Robert (2004) Easy Reflow. *Circuit Cellar,* **168** (available online at www.circuitcellar.com)

Rooks, Bob, (2006) "Screen Print and Reflow SMT Boards at Home," http://www.pcbexpress.com

Stencils Unlimited (www.stencilsunlimited.com): Solder paste, laser-cut prototype stencils for solder paste application, and information on using a toaster oven for reflow soldering.

Mouser Electronics (2006) (www.mouser.com): An electronic distributor with a very large selection of parts, excellent customer services, and favorable terms for small orders.

PCB Express (2006) (www.pcbexpress.com): Prototype PCBs (including multilayer) and prototype stencils. Will provide up to $350 per year of PCBs per year to non-profit educational institutions. Interesting links to other information sources

Mini Micro Stencil Inc. (2006) (www.minimicrostencil.com) Flip-up rework stencils (http://www.minimicrostencil.com/flip-up_stencils.htm).

LPKF Laser and Electronics (2006) (www.lpkf.com): Manufacturer of CNC milling machines

T-Tech Inc. (2006) (www.t-tech.com): Another manufacturer of CNC milling machines for prototype PCB fabrication.

Think & Tinker (www.thinktink.com): PCB prototyping equipment and information. Inexpensive tools for CNC milling machines.

Homebrew_PCBs (2006) (http://groups.yahoo.com/group/Homebrew_PCBs/): An internet group dedicated to "Designing, making, etching, soldering, printed circuit boards yourself".

The American QRP Club (2006) (www.AMQRP.org): a source of information on the SoftRock40, as well as other construction projects with emphasis on low-power and portable amateur radio equipment.