

## **2006-2007: RAPID PROTOTYPING OF PRINTED CIRCUIT BOARDS USING AN ENGRAVING LASER**

### **Jeffrey Mountain, University of Texas-Tyler**

JEFFREY MOUNTAIN obtained the Ph.D. degree from the University of Texas at Arlington. Dr. Mountain's interests are in mechatronics, robotics, automation, solar energy applications and CAD/Graphics. He also has experience in the design and installation of mechanical building systems, specifically plumbing and HVAC systems. Instrumentation systems design and electrostatic powder coating system applications round out his areas expertise. He also holds a current State of Texas master plumber's license and is active in the area of K-12 outreach. Dr. Mountain served on the faculties of the University of Texas at Arlington, University of Arkansas at Little Rock and the University of Illinois at Urbana-Champaign prior to joining the UT-Tyler faculty.

### **David Beams, University of Texas-Tyler**

DAVID M. BEAMS is an Associate Professor of Electrical Engineering at the University of Texas at Tyler. He received his BS and MS degrees from the University of Illinois at Urbana-Champaign in and the Ph.D. from the University of Wisconsin-Madison. He has had over 16 years of industrial experience in addition to his 8 years with UT-Tyler. He is a licensed professional engineer in Wisconsin and Texas and holds or shares four patents.

# Rapid Prototyping of Printed-Circuit Boards with an Engraving Laser

## Abstract

Lack of a good system for rapid prototyping of printed circuit boards has been a major bottleneck in Senior Design at the University of Texas at Tyler. A simple method using artwork printed on toner-transfer paper and transferred through heat and pressure to copper-clad circuit-board stock had been successfully applied to simple designs but showed itself to be unsuited to the complexity of printed circuit boards developed for senior design projects. This unsuitability was due to two factors; low density (due to inability to accurately control line widths) and highly-variable results.

Alternative methods have their drawbacks. Tooling charges for commercially-produced circuit boards are expensive if only a small number of examples are to be built. Inexpensive prototype printed-circuit boards can be obtained from specialty circuit board vendors, but these vendors usually require the use of their proprietary software. Small high-speed milling machines that can cut circuit traces are available, but it is difficult to justify the expense of a dedicated circuit board mill for annual production of a few circuit boards.

An alternative being investigated by the University of Texas at Tyler is the use of a CO<sub>2</sub> engraving laser as the principal patterning element in a system for rapid prototyping of printed circuit boards (PCBs). The engraving laser was acquired principally for research into microfluidic devices, but it appeared that it might be an effective way of patterning masks for chemical etching of copper-clad circuit board material. Efforts have so far focused on ablating a thin film of a resist material, leaving copper cladding exposed to chemical etching in the areas where the resist was ablated. Marking, scoring, and drilling with the laser have also been attempted. This paper describes the methods and accomplishments to date as well as directions for future work.

## Prior experience: toner-transfer process

There is a broad literature aimed at electronic hobbyists which describes methods for transferring laser-printer toner to copper-clad PCB material as an etch-resist material. (A cursory search for “toner transfer process” with an internet search engine revealed over 420 “hits,” of which the majority appear to involve fabrication of prototype PCBs). A common version of this method involves printing a PCB artwork pattern with a laser printer on transparency film and using a household iron to transfer the image to PCB material. Commercially-available toner transfer paper is available (Pulsar, Clearwater, FL) to be used in lieu of the transparency film<sup>1</sup>. The artwork is printed with a laser printer on the side of this paper that is covered with a water-soluble starch. Heat and pressure are used to bond the toner and paper to the PCB, and the paper is lifted away (leaving the toner fused to the PCB) by soaking the board and paper in water.

Experience with the toner-transfer process at UT-Tyler gave inconsistent results. A “recipe” involving time, temperature, and pressure was developed by experimentation with a heated-

platen press (Carver Inc., Wabash, IN), but the results varied from one attempt to the next. It was surmised that the chief variable was pressure, which could not be measured with any degree of accuracy with the pressure gauge of the press. It was not uncommon to have to strip the toner from the PCB and start over with another toner-transfer sheet because part of the artwork failed to fuse properly to the PCB (probably due to insufficient pressure). It was also not uncommon to find traces that had “bled” into each other (probably due to excessive pressure). The toner image was also found to be porous, and large areas of toner (meant to cover ground planes or wide traces) allowed extensive pitting during the etch process. The vendor of the toner-transfer paper does provide a suite of equipment for this process, and claims very fine definition (0.006in line widths), but such results apparently depend upon having their specific equipment and PCB material.

There were also problems associated with accurate registration of pads and holes. If holes were drilled first, the toner-transfer process was not as reliable as when applied to an undisturbed surface. Since the drilling operation would potentially dislodge the toner, the best patterns were obtained by drilling after the etching process was completed. Unfortunately, it was difficult to achieve accurate alignment in the subsequent CNC milling process used to drill the holes. Minor errors in both location and orientation usually were not evident until the board was completed, resulting in an unusable prototype.

### **Other methods for printed circuit board fabrication**

Other options for creating one-of-a-kind printed circuits include contracting with a prototype circuit board fabricator. There are numerous fabrication service companies available. A simple internet search for “printed circuit board fabricators” resulted in 696 “hits.” Each company has its own design rules, some requiring multiple files in several different formats. Many have relatively quick processing time and the prototyping fees typically start around \$50. This type of service does not work well in a typical university environment. The only way to minimize the number of different design rule combinations is to have a sole source for PCB fabrication, which may require significant trial and error before a suitable vendor can be identified. There are also the problems associated with university purchasing rules, with which undergraduate students are not familiar.

Another method for on site fabrication involves the use of specialized, engraving, routing or milling machines; also referred to as “mechanical etching”<sup>2</sup>. Typically, a system comprised of specialized software and hardware is required to accomplish the PCB fabrication process. Although there are several manufacturers of mechanical etching equipment, LPKF Laser & Electronics<sup>3</sup> and T-Tech Inc.<sup>4</sup> are considered the major manufacturers. While these systems are very capable of producing viable, one-of-a-kind PCBs, their single-use function and special use translation software made them less desirable for our intended purpose. It was difficult for a small program to justify this type of expense for a dedicated system that will typically produce at most 10–15 prototype boards per year.

### **Description of the laser prototyping process**

The process involves computer-aided design (CAD) software to create “laser ready” artwork. The laser “prints” directly on a blank PCB, eliminating the need for an optical mask (as in photolithography) or for a one-use toner mask (as in the toner-transfer process). A commonly-available acrylic coating is used as an etch resist on the copper side of the PCB; the laser ablates (burns away) the acrylic coating in regions where copper is to be removed from the finished board. A separate ablation step is used to “drill” through-holes and to mark the component side of the PCB. The masked and “drilled” board is etched with ferric chloride solution to remove unprotected copper. The etch resist is stripped using an organic solvent (xylene or acetone), and the uncoated copper is tinned using a liquid-tinning solution. Figure 1 is a flow diagram of the circuit board prototyping process using the laser. Details of the process elements follow.

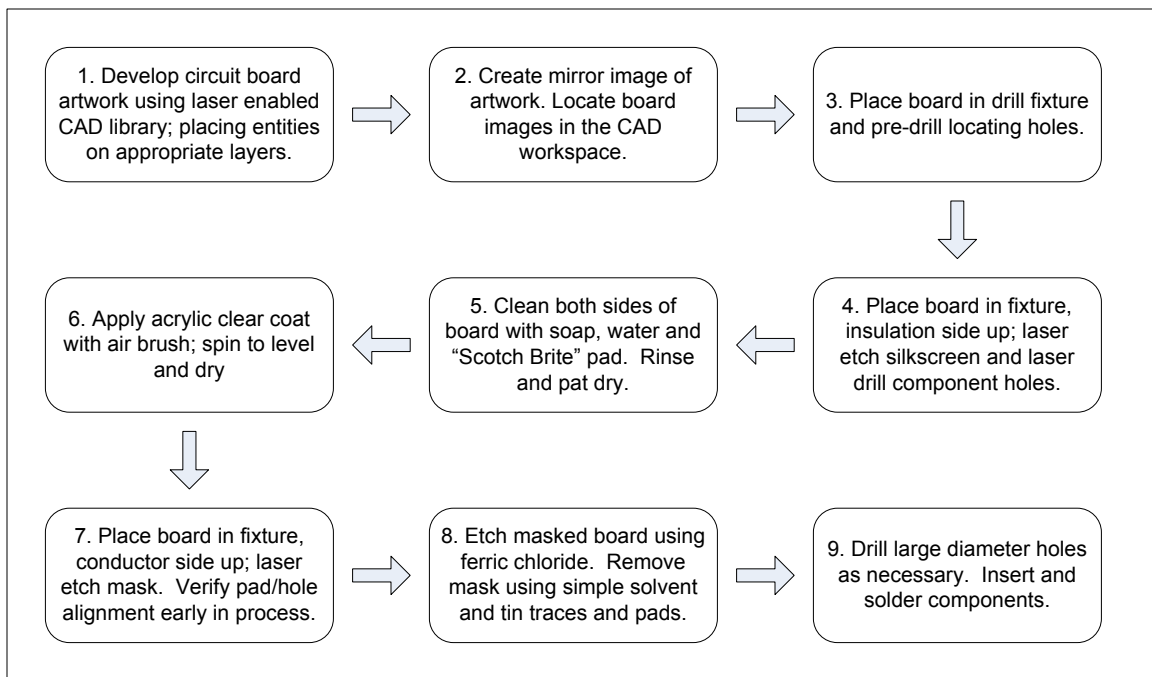


Figure 1. Process flow diagram for laser process prototyping of printed circuit boards.

### Engraving laser

The laser used for this patterning process is a CO<sub>2</sub> engraving laser manufactured by Universal Laser Systems (ULS) of Scottsdale, AZ. The model X2-660 is configured with twin air-cooled 60W laser tubes which can combine to produce an effective output power of 120W. The laser chassis accepts any of the Universal Laser Systems air-cooled laser modules ranging from 10W to the 60W tubes used on this particular system. This laser may also be configured as a single-tube 60W laser by disconnecting one of the tubes.

The laser manufacturer provides a software driver that causes the X2-660 to appear as simply another printer device to a personal computer running a Windows operating system. This driver provides the ability to convert CAD, graphics, or text-based data into motion and laser power commands.

The laser uses stepper motors to produce 0.001-in (0.0254 mm) motion resolution in both *x*- and *y*-axes. Laser power is adjusted using a pulse width modulation (PWM) strategy. Image resolution can be adjusted in two ways: by an Image Quality software setting that varies resolution in dots per inch (dpi), and by selection of the final optical lens to adjust laser spot diameter (and ultimately laser power density at the board surface). Additional laser control is achieved by color and entity coding. Both raster and vector graphics are supported and are used to produce different engraving effects. Vector-based entities (lines, arcs and circles defined with line width attributes of approximately 0.001 in) are interpreted as point-to-point vector graphics and place the laser into an always-on, cutting mode. Other graphics are interpreted as raster-based bit maps and are reproduced dot-by-dot as controlled by the Image Quality setting. Laser mode (raster, vector, raster and vector, or skip), motion speed, laser power, and the number of laser pulses per inch (PPI) are assigned based on the entity color. During any single “printing” cycle, the following eight colors may be used to define the laser characteristics: black, red, green, yellow, blue, magenta, cyan and orange. The ordering of the processing scheme appears to be that raster operations occur before vector processing and that colors are evaluated in the order previously listed. These settings are adjusted using a Windows-based laser settings dialog box accessed by selecting the properties of the X2-660 laser. The interface is relatively easy to use and setting combinations can be saved and recalled as named settings files; greatly facilitating the multi-purpose role of this laser.

The final optics lens selection plays a significant role in the applied power and image quality of the circuit pattern. The standard optics have a 2-in (50.8mm) focal length and produce a 0.005-in (0.127mm) diameter spot at the focal point and have been chosen for this application. ULS has developed an optional High Power Density Focusing Optics (HPDFO) system that produces a 0.001-in (0.0254mm) diameter spot with a higher power density. The laser was acquired primarily for the fabrication of microfluidic devices, and the optional lens was purchased for that application. The HPDFO optics were initially tested in the PCB-patterning application and produced excellent initial results. But the HPDFO optics are an expensive option that the authors determined was not mandatory for this application. All testing and results data reported in this paper were obtained with the standard (2-in focal length) optics.

### **CAD program**

Although graphics programs such as Corel Draw are more commonly used in the engraving industry, the laser manufacturer does support AutoCAD as a graphics-creation application. At the University of Texas at Tyler, we have adapted the Draft component of Solid Edge, a UGS CAD software product, as our patterning environment. Although other software packages have not been tested, it is believed that any graphics software could produce the direct print results that we have obtained if that software provides object color control, precise line-width control, ability to hide and show entities by layers, and basic editing commands (such as mirroring an image about a specified axis). The X2-660 user interface limits entities to the eight primary colors previously listed, but the ability to save and recall setting profiles, combined with the previously described CAD software attributes, can be used to achieve a variety of effects.

### **Laser-ready Artwork**

Although the laser hardware and interface system have been an enabling technology, artwork that contains the proper geometric entities is required to minimize the trial and error efforts of students.

The laser engraving system presented an opportunity not only to define conductor patterns on copper-clad PCBs but also to drill and to mark (“silkscreen”) the unclad side of single-sided PCBs. The ability to drill PCBs with the laser is particularly attractive, because the process for machine-drilling of prototype PCBs was rather complex, involving the following additional steps: 1) isolating center-point objects for all holes in a separate file; 2) exporting the center-point locations using the AutoCAD DXF standard; 3) importing the resultant .dxf file into our resident CAM software; and 4) creating the CNC file for drilling holes. All holes were drilled to the diameter of the smallest holes; holes of larger diameters were only “center punched.” Larger holes were always re-drilled by hand because there were relatively-few larger-diameter holes, and it was less effort to drill them manually than to create separate CNC files for each size of the larger-diameter holes. To maximize the CNC drilling capabilities, all holes were drilled to a diameter of 1/32”, which would accommodate a 1N4001 diode lead, but was slightly oversized for most chip carriers and component leads. The oversized holes made component insertion easy, but the soldering process more difficult due to the larger voids around most components.

The ability to avoid the data-isolation and file-translation steps, while gaining the ability to drill holes of various diameters in a single step using the laser seemed worth investigating. Although a CO<sub>2</sub> laser in this power range does not appear to be capable of piercing standard one-ounce copper cladding (presumably due to its high thermal conductivity and reflectivity), it was discovered experimentally that sufficient heat could be concentrated on the unclad side of the PCB to burn through the PCB material, including the cladding. Several geometry patterns were tested for drilling, including concentric circles, a synthetic spiral pattern, and multiple center-point objects. Different speed and power settings were also tested to minimize the scorching associated with the ablation of the PCB material (fiberglass-epoxy composite) while making holes large enough to allow component leads to be inserted with relative ease. Drawing multiple vectors in the same location will cause the laser to repeat the pattern as each vector will be sent to the laser individually. These additional vector entities would not be visible on a computer screen or in a raster-scanned image.

A combination of concentric, co-existent circles, combined with multiple center points, provides the necessary geometry to drive the hole-drilling operation. At present, hole geometry sized for chip carriers is embedded into elliptically-shaped pads configured for a variety of commonly-used components. Additional sets of geometry are embedded into “through pad” entities that allows for the larger-diameter leads associated with 1/4W resistors and diodes. This drilling process is not suitable for double-sided PCB material, so a single point entity on a special “drill” layer located at the center of every pad makes the layout process compatible with double-sided boards. These point entities can be subsequently exported to produce a CNC drilling pattern using the extended process previously described.

Layered data for marking (or “silkscreening”) the component side of the PCB, as well as for identifying the pin functions, has been included for both pedagogical and application-related reasons. Figure 2 illustrates the range of data and the layering scheme that has been developed

for this application. This Pin\_name layer is typically suppressed once the circuit has been fully developed.

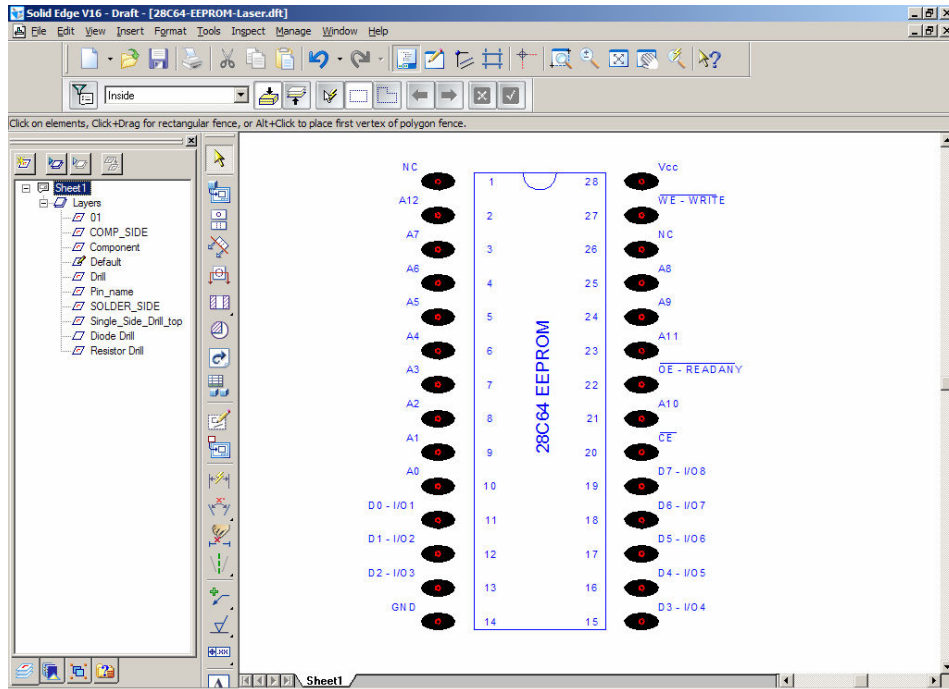


Figure 2. Component library element shown in the CAD environment with associated layer list.

Control of laser operations for any object is linked to the object's color. A procedure was developed to produce artwork that was viable for both the laser ablation process and for printing to a standard printer for documentation and verification. A prototype drawing defines the 32 in. × 18 in. laser workspace. The user initially defines the board outline and two indexing holes on the right-hand side of the board. In the CAD environment, these holes represent physical fixturing holes and will define the line of symmetry for producing the mirror image artwork. Components are added from the library and copper traces or conductor planes are defined on the appropriate layers. This completes the first step of the prototyping process.

Once the board outline and the registration holes have been defined on a Board Outline layer, a fill pattern is placed on a Board layer. A contrasting color is used for this fill pattern, we have selected yellow. This layer is then hidden until the unmasking operation is performed. At that point, components are placed on the board and traces are drawn on the SOLDER\_SIDE layer, using black as the entity color. Line widths are adjusted as necessary to provide adequate conductors. Although the widths are not always obvious, or accurately portrayed, on the computer screen, the effects of line width specification are obvious when either printed to a standard printer or sent to the laser engraver. We have found that 0.5 mm (0.02 in) line widths work for most applications, but that 0.35 mm (0.014 in) line widths may be used if attempting to pass two traces between a set of pads. Most student project boards should not require using double trace passes, but the capability is there if necessary. Solid Edge readily supports single

line widths up to 2 mm (0.079 in). Any line width may be specified, although larger widths may produce unexpected effects at the end points. Drawing a closed region and using a fill pattern is the best method to define larger conductor areas.

### The board production process

After the artwork has been defined, completing the first step in the process, the final steps in the board production process can be undertaken. To begin step two, the mirror image is created on a separate drawing sheet by initially copying all of the original board geometry. A “Save As” option can be used as well, creating a copy of all the geometry. A vertical construction line is drawn through the centers of the two indexing holes. All of the geometry is then selected and mirrored about a horizontal construction line starting at the midpoint of the previously drawn vertical line. Only the Board, Board Outline and SOLDER\_SIDE layers should remain on in this drawing since it represents the conductive layer mask geometry. The mirrored geometry now represents the conductor side of the board as seen by the laser during the masking process.

Returning to the original drawing, all layers except the Board Outline, Component and various Drill Layers should be hidden. The top right hand corner of the board must be moved to a Reference Point location identified on the prototype drawing. This location represents the top right corner of an indexing fixture. The actual location was determined experimentally and varied slightly depending upon the final focusing optics used. The level of precision needed for this “mirror-image” process is not typical of the engraving industry applications, and this procedure has demonstrated excellent repeatability.

The CAD based fixture holes must conform to pre-defined locations corresponding to a simple drilling jig, shown in Fig. 3. To complete the third step of the process, a jig was created to produce reliable indexing and registration of the board. These holes are easily drilled by hand using the jig. The jig can accommodate a board with up to a 6 inch edge dimension.

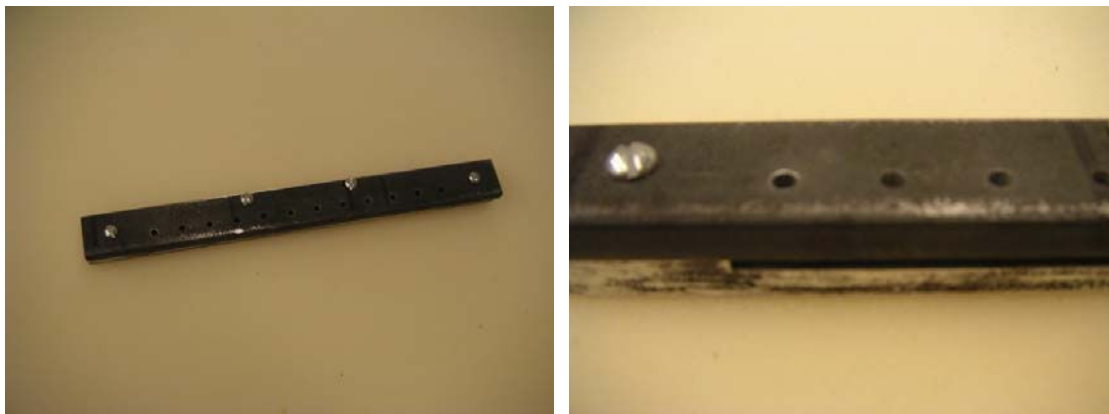


Figure 3. Drilling fixture for indexing holes for copper-clad circuit boards of widths up to 6 in (15.24cm).

Steps four through six begin with initially mounting the target PCB on the indexing fixture with the unclad side up. The holes and the component markings are then created by the laser. Figure 4 illustrates the marking process, showing a fixture-mounted PCB before and immediately after



the initial laser processing. Figure 5 shows the circuit board after cleaning the surface with soap, water and a “Scotch-Brite” abrasive pad to remove the char residue. Following the initial ablation and cleaning processes, a resistive coating is applied to the copper side of the board. Currently, we are using a clear coat acrylic modeling paint, applied with an airbrush and immediately spun to both level and dry the material. A black pigmented acrylic coating was initially tried because ablation of this material left a readily-visible mask pattern, but experiments showed that the black acrylic left behind a carbon black film residue that inhibited etching.

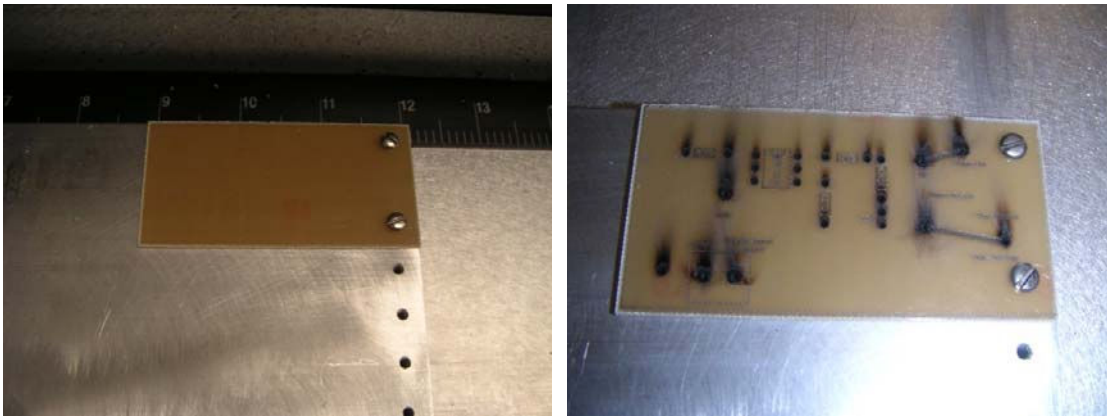


Figure 4. A fixture mounted circuit board before and after initial laser drilling and marking.

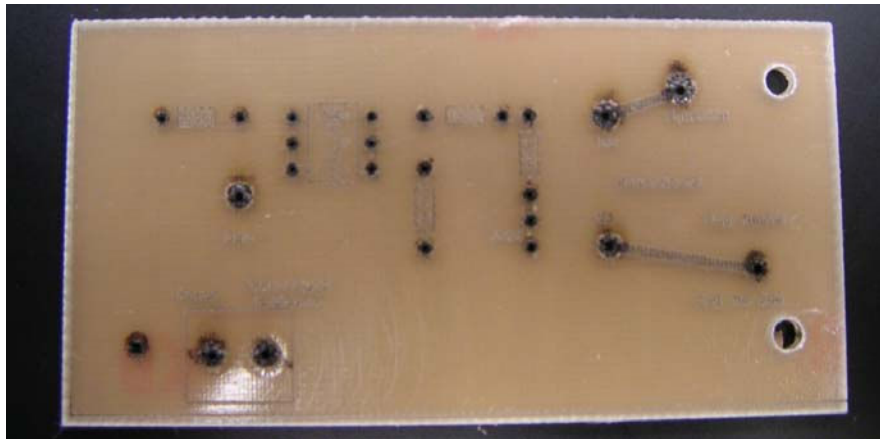


Figure 5. The completed component side of a circuit board after final char removal.

To complete the process, steps seven through ten, the masked PCB, it is flipped about the drilled indexing holes and remounted into the laser with the copper-clad side up. The mirrored, conductive-layer mask geometry is then downloaded to the laser, using a raster scan ablation of the board area that has not been obscured by traces. The ablation process is monitored to verify proper registration of the pad masks on the holes. If an alignment problem becomes apparent, the laser processing can be interrupted and the board quickly remasked and realigned. The copper-side ablation process takes about 30 minutes for a typical 4 x 6 inch board, regardless of chip density. If only a “trace cutting” ablation is required, processing time may be significantly reduced, but that reduction may be easily offset by the time required to define the more limited geometry. The hole drilling and marking operation is relatively rapid, but the time required is

dependent on the number of holes and markings. Processing times for the 1½ in × 3 PCB shown in Figure 6 were 5 minutes, 40 seconds to drill/mark the component side and 7 minutes, 26 seconds to process the conductor side of the board. Cleaning and application of the acrylic mask took approximately 10 minutes. Figure 6 shows a mounted circuit board immediately following the masking process, which leaves a residual ash that is easily removed using soap, water and a paper towel. The board is then chemically etched, tinned, and populated with electronic components. Figure 7 shows a triac control circuit constructed on the PCB.

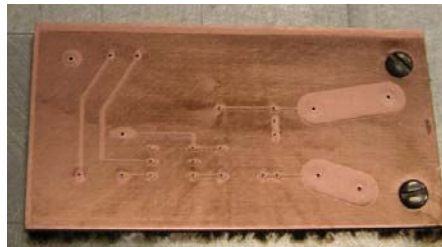


Figure 6. Fixture mounted circuit board immediately following laser ablation processing.

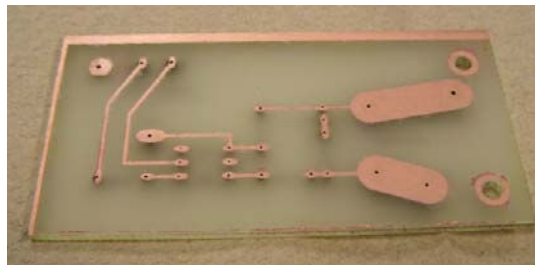


Figure 7. The sample circuit board following the chemical etching process.

## Results

The overall results have proven to be considerably more reliable and repeatable than the toner-transfer process and take advantage of the capabilities of an existing piece of equipment. We have developed simple tooling that allows students to create boards up to 9 in × 6 in (22.9cm × 15.2cm) in size. Larger boards can be accommodated in the laser engraver, but would not work with other tooling created for this project. The process is repeatable and uses only chemicals (clear acrylic hobby paint, ferric chloride etching solution, acetone or xylene, and liquid tin-plating solution) that are readily available and which may be stored and handled with simple safety precautions. Figure 8 shows a completed (etched, tinned, and populated) board. Six of the holes required additional drilling (one screw mount and five power bus connections), but this task was readily accomplished using a hand drill.

The initial investment in this particular laser system was significant, but tests have been conducted at significantly reduced power to judge whether a smaller, lower-power, freestanding or desktop laser (such as the Universal Laser Systems M-360 or VL-300) could be used. Initial results indicate that a 50–60W laser should be capable of drilling the holes, and would certainly be capable of performing the mask-preparation process. These systems are available at a significantly lower cost than the X2-660 used in our work. Although the process has not been

attempted on a double-sided board, we expect that the consistent quality of the final product, due to accurately registered masking on both sides of the board, will greatly reduce the time and effort to produce a one-of-a-kind double-sided board.

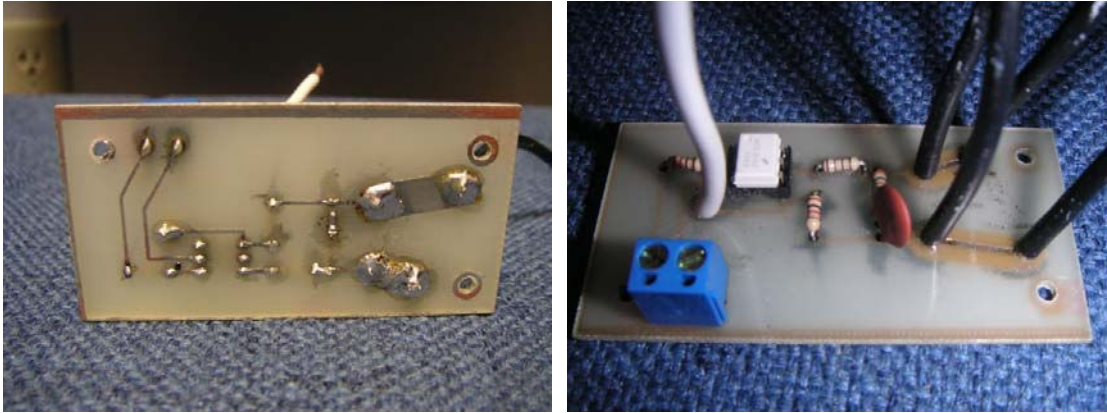


Figure 8. Completed triac driver board.

## Conclusions

A prototyping process using an engraving laser has been developed that shows promise to streamline the process of making small quantities of prototype PCBs. It has been tested in the fabrication of small single-sided PCBs; we plan to extend the process to larger boards and to double-sided PCBs. A major test of the capabilities of this process will take place in spring, 2006, as we expect the process to be applied to senior-design projects.

## Bibliography

1. Pulsar (2006), *Instant printed circuit boards in fifteen minutes with the Toner Transfer System*, <http://www.pulsar.gs/PCB/index.html>, (17 January 2006).
2. PCB Milling (2001), PCB Milling Homepage, <http://www.pcbmilling.com>, (16 January 2006).
3. LPKF Laser & Electronics (2005), PCB Quickturn Prototypes, <http://lpkfusa.com>, (16 January 2006).
4. T-Tech Inc. (2005), T|Tech Quick Circuit Prototyping Systems, <http://www.t-tech.com>, (16 January 2006).