PROJECTS THEORY APPLICATIONS CIRCUITS TECHNOLOGY

NUTS AND VOLTS

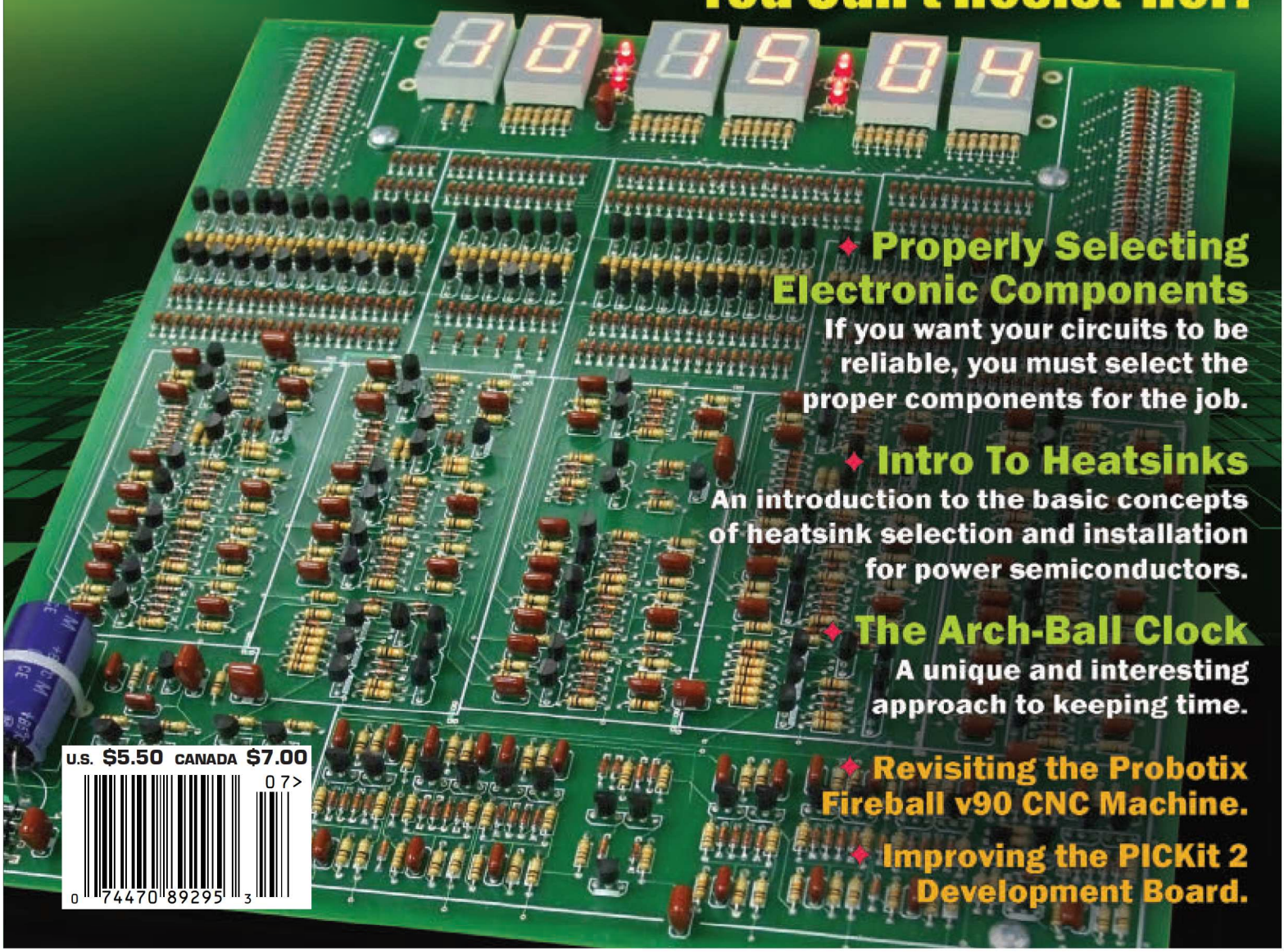
NUTS AND VOLTS

www.nutsvolts.com
July 2009

EVERYTHING FOR ELECTRONICS

BUILD THE Transistor Clock

You Can't Resist-Her!



Properly Selecting Electronic Components

If you want your circuits to be reliable, you must select the proper components for the job.

Intro To Heatsinks

An introduction to the basic concepts of heatsink selection and installation for power semiconductors.

The Arch-Ball Clock

A unique and interesting approach to keeping time.

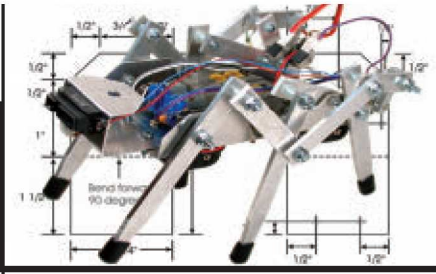
Revisiting the Probotix Fireball v90 CNC Machine.

Improving the PICKit 2 Development Board.

U.S. $5.50 CANADA $7.00

07>

0 74470 89295 3



■ BY VERN GRANER

THE PROBOTIX FIREBALL V90 CNC VISITED

HAVEN'T WE BEEN HERE BEFORE? If the machine shown here looks familiar, it's because we first introduced you to it back in the December '08 column. In that article, we detailed the PROBOTIX Fireball v90 — one of the first high-accuracy/sub-$1,000 CNC systems on the market. We explored what came in the package, detailed how the machine went together, and gave a few examples of how the machine could be used. We then shared our experiences as newbies using the machine in a high-pressure environment for a couple of non-stop days at Maker Faire Austin. At that event, the machine was used to cut recycled CD-ROMs into gears, key chains, and even a set of snazzy earrings (Figures 1 - 3)!

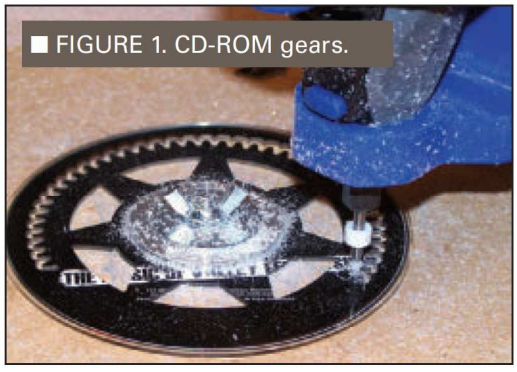
Since then, we've had time to become more familiar with the v90 and — though there's nothing wrong with cutting CD-ROMs into earrings — we figured it would certainly be more practical to use the machine to create something useful for the electronics hobbyist. This month, we're going to describe how our v90 has evolved since its debut and then show how the machine can be put to some real work making practical items for use in your every day electronic and robotic projects.

READY TO RUN? YES!

When we originally received the V90 back in August '08, the hardware came with all the required electronics components, but you were expected to provide (or create) your own electronics enclosure. In addition, you had to solder/heat shrink the wires to each of the stepper motors and then attach the ends of the wires to each of the motor controller boards using screw terminals. Though this approach worked fine for us and is still a viable option for folks who expect their CNC system to "stay put," having the wires screwed to terminals in our system didn't lend itself to portability.

Any time we needed to move the machine, we had to unscrew each wire from the stepper controllers, pull them out of the hole in the enclosure, and then set the control electronics inside the enclosure for transport (**Figure 4**). When reassembling, we had to reverse the process and make sure we got all the stepper motors wired back to their controllers correctly. Also, using our method of electronics assembly (i.e., screwing everything down to a hunk of wood), you have to take special care to avoid both the spinning fan blades and possible short circuits with the exposed electrical connections (**Figure 5**).

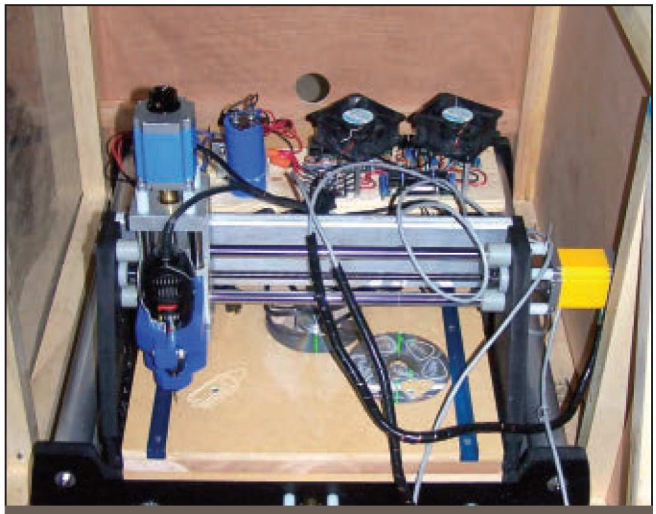
To avoid all this hassle, a new "Ready To Run" option allows you to purchase all the electronics factory assembled in a nice, fan cooled custom cabinet (**Figure 6**) with



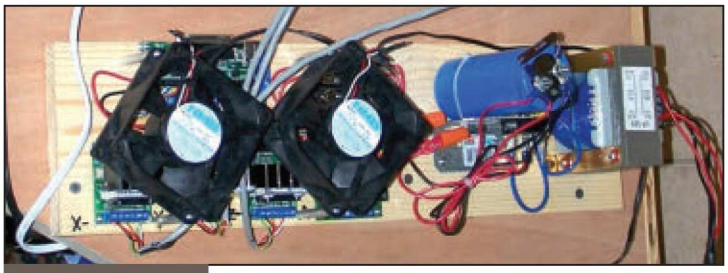
■ FIGURE 1. CD-ROM gears.

■ FIGURE 2. CD-ROM cut into shapes for key chains.

■ FIGURE 3. At Maker Faire, a young passerby turned a pair of Texas key chains into earrings.



■ FIGURE 4. The v90 ready for transport after disconnecting the stepper motors.



■ FIGURE 5. The v90 power supply, motor controllers, and cooling fans.

■ FIGURE 6. The Ready To Run case contains the power supply and the motor controllers.

clearly labeled DIN style connectors on the back (**Figure 7**) and the power supply and associated board enclosed professionally inside (**Figure 8**). This certainly makes set-up a snap and avoids both exposed electronics and stepper mis-wiring issues as the motors with the Ready To Run kit come with the DIN plugs pre-attached (**Figure 9**).

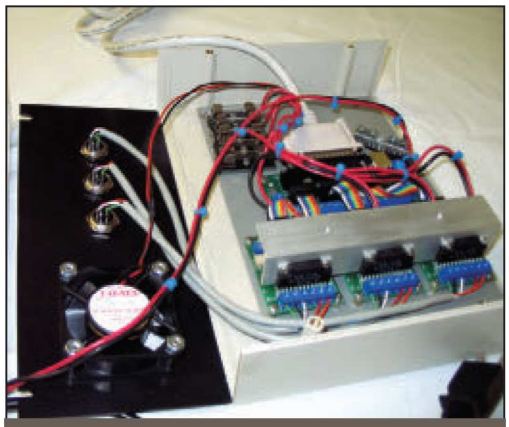
JUST LIKE ONE OF THE FAMILY

One of the nice things about a product like the v90 is the community that grows up around it. There is a very active and helpful group of folks on the PROBOTIX mailing list, in fact, even Len Shelton (the owner of PROBOTIX) is known to make frequent appearances to answer questions. Through reading the list and having discussions with other owners, we graduated from a Dremel rotary tool, to a Porter Cable router, and finally worked our way up to a Bosch Colt. Upgrading tools was a simple process as PROBOTIX makes various tool mount adapters to help facilitate experimenting with different cutting tools, including one for the Colt (**Figure 10**) which we currently have on our system.

Without getting too involved in the process of moving up through the various cutting tools, the bottom line was the Bosch Colt displayed the least amount of runout. "Runout" is basically the amount that the cutting bit deviates from a perfectly circular, perpendicular rotation. Excessive runout in your cutting system can lead to broken bits, inaccurate cuts, and other undesirable results. (For more details on runout, see the **Resources** section.) The last piece of the puzzle (in regard to the mechanical aspect of the system) is that when you are asking the machine to perform a job, it's crucial to have the right bit for that job! There are literally hundreds of different styles and sizes of bits, and it helps to have expert advice to both purchase the right part and to ensure a long life for that part. After buying (and breaking) quite a few bits from discount sources, Len suggested we speak with the folks at **PreciseBits.com**. After reading their website, we ordered bits for the two things we really wanted to do: making circuit boards and cutting case bezels. So, with our v90 converted to the new Ready To Run kit, the Bosch Colt router firmly in the grasp of the z axis, and fitted with some shiny new bits from **precisebits.com**, it seemed it was time to spin up the router and cut some stuff ... or was it?

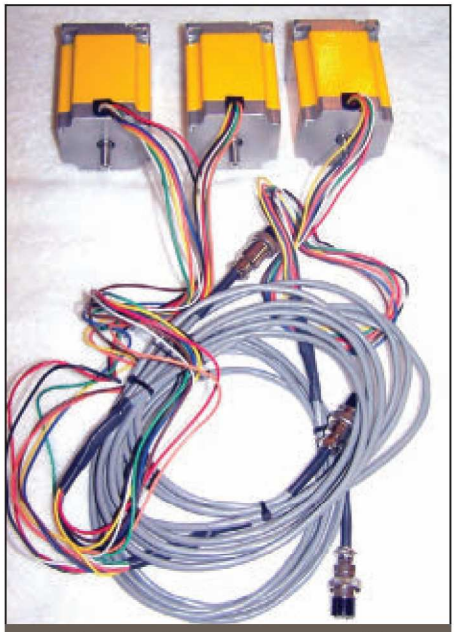
THE EAGLE HAS LANDED

Before we fire up the router and start making dust, we have to begin by opening up the computer and dealing with some software first. There are many paths to get from



■ FIGURE 8. Inside the Ready To Run case.

■ FIGURE 7. Rear view of the Ready To Run case.



■ FIGURE 9. Stepper motors with wires attached and terminated with DIN plugs.

■ FIGURE 10. The Bosch Colt router in the PROBOTIX Fireball v90.

your design to a finished piece (see resources for software choices), but we typically use EAGLE if we intend to use the resulting layout to cut a printed circuit board (PCB) using the v90. Available as a free download from CadSoft, EAGLE is an easy to use software tool that can be enhanced with plug-ins — one of which is particularly important to using a CNC machine to create a PCB (we'll get to that in a bit). Before you commit to a commercial PCB order, it's a good idea to create a prototype of that PCB and the v90 makes this very simple. As a real-world example, we'll show a step-by-step on how to create the prototype of the Das BlinkenBoard from last month's column.

THE SCHEMATIC BONE'S CONNECTED TO THE ...

Here's a very quick overview of how we start with EAGLE and end up with a circuit board ready to solder up. We begin by drawing up the schematic in EAGLE. Then, we create the PCB by clicking the "board" icon (looks like two gates below the "View" icon in the tool bar. A pop-up message asking if we want to create the PCB from the schematic is presented, to which we answer yes.

EAGLE now creates an empty PCB with all the parts off to the left of the board. All connections are shown as yellow lines (which are elastic). As you move parts around, the lines stay connected.

Spend some time arranging the parts, trying to minimize crossing lines by rotating the parts. (TIP: Right-click while a part is selected and it will rotate 90 degrees with each click). Once you have a layout you like, it's time to auto-route the board. While there are a lot of parameters that can be set, we'll do this with the defaults. The auto route button is under Tools.

EAGLE auto-routes and (hopefully) shows 100% complete in the bottom left corner. If the board is too full or if the routing rules (a.k.a., Design Rule Checks) are too restrictive, your design may not route on the first try. It's fairly simple to see what needs your attention as un-routed traces will still be yellow. In our example here, the routing completed to 100%.

Even after the auto-route is complete, you may move some traces to suit your preferences using the four-arrow select tool on the left tool bar. For example, we increased the thickness of various traces where possible ("change width" using the wrench tool on the left tool bar).

ISOLATING — THE NEXT STEP

The next step involves the plug-in for EAGLE (also know as a ULP or User Language Program) I mentioned earlier. It's called PCB-GCODE and is a free download thanks to John Johnson at **pcbgcode.org** (see Resources). PCB-GCODE converts the EAGLE layout from a standard PCB layout to an "Isolation" type layout.

An "Isolation Route" is where we remove only enough copper to isolate the traces on the board from one another. This looks different than a normal PCB which only retains the necessary copper to make it work. As a plug-in for EAGLE, PCB-GCODE is a bit intricate to get going, but the following steps should be helpful.

To get PCB-GCODE started, download and install it on your Windows machine using the instructions included on the website. Once PCB-GCODE is installed, type "run" in the command line of EAGLE (**Figure 11**), then type "PCB." You should see "pcb-gcode_setup.ulp" and "pcb-gcode.ulp." Select the "pcb-gcode_setup.ulp"

RESOURCES

- PROBOTIX Fireball v90 in December '08 issue **http://nutsvolts.texterity.com/nutsvolts/200812**
- PROBOTIX **www.PROBOTIX.com**
- PROBOTIX Fireball v90 at Maker Faire Austin **http://makerfaire.com/pub/e/2151**
- Measuring runout tutorial from PreciseBits.com **www.precisebits.com/tutorials/spindle_runout.htm**
- Software paths discussion **www.PROBOTIX.com/cnc_software**
- Fireball discussion boards: **www.FireballCNC.com** **http://groups.yahoo.com/group/Fireballcnc**
- EAGLE from CadSoft: **www.cadsoft.de/freeware.htm**
- PCB-GCODE plug-in for EAGLE **http://pcbgcode.org**
- Vectric VCarve Pro **www.vectric.com**
- Das BlinkenBoard **www.DasBlinkenBoard.com**
- The Robot Group **www.TheRobotGroup.org**

to run first (**Figure 12**).

Start in the Generation Options tab by changing the parameters to match the board thickness (milling depth), clearance minimums and maximums (isolation default/maximum), and the etching tool size (**Figure 13**). Remember to un-check the "Generate top outlines/drills" as we are doing a single-sided PCB with traces on the bottom.

Next up is the Machine tab. You can modify the parameters to match your CNC machine including Spin Up Time and Drill Dwell (in seconds), drill depth (I used 0.070" for 0.064" copper clad), and tool change position in x, y, and z (the CNC will move to this position for a tool change). The GCODE Style tab allows you to select the appropriate version; in our environment, we chose EMC for EMC2 under Linux.

After making these changes, choose "Accept and make my board." You should see a pop-up asking to run Windows' viewer (**Figure 14**). After accepting this, there should be a series of drawings, each followed by a "close this window" dialog box (**Figure 15**).

When done, the PCB-GCODE plug-in will generate etch, drill, and mill files, typically located in the "My Documents/Eagle" folder on your machine. The "etch" file contains the mechanical etching of traces and pads; the "drill" file contains the holes to be drilled; and the "mill" file contains the perimeter cut information for the finished board.

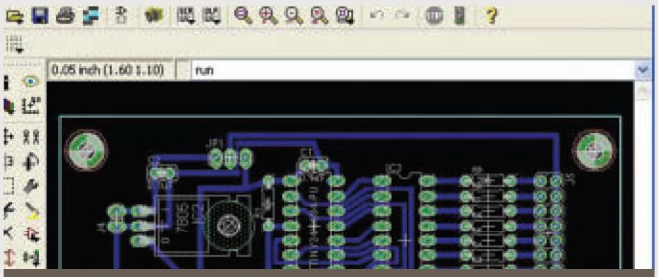
HERE THERE BE PENGUINS!

At this point in our environment, we simply copy the ".ngc" files to a "thumb drive" and take that over to the Linux machine we have connected to the Fireball v90. Once the files are copied, a few small changes need to be made before we start. First, the drill file has to be edited for tool changes. Look for lines that start with M06 and put everything from the semi-colon to the end of the line in parentheses. For example, this: "M06 T01 ; 0.0320" should become this: "M06 T01 (; 0.0320)" This change will make EMC2 stop and ask for a tool change.

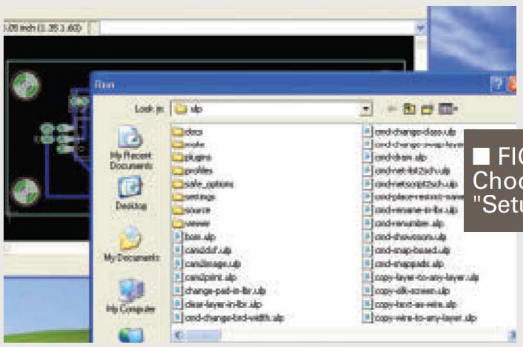
Setting the depth for PCB etching may be a bit tedious. Typically, you start with the z axis "zeroed" on the surface of the copper clad (this can be done by eye or by placing a small piece of paper under the bit and then owering the z axis until the paper is pinned. Then, back

A THANK YOU ... OR TWO ... OR MORE ...

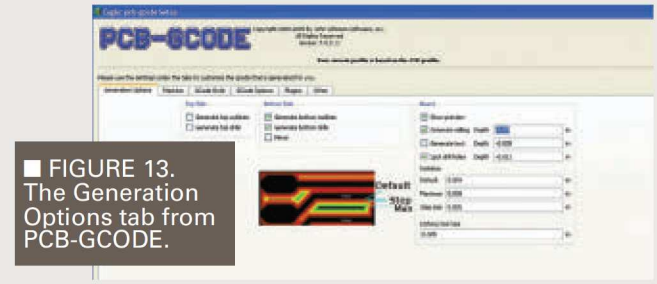
I want to thank Paul Atkinson for all his help in preparing the material for this article and his wife Dolly for letting Paul come out and play (thanks for the pizza too!). Also, special thanks go out to Len Shelton of PROBOTIX for his unflagging dedication to the vision of home-CNC and his top-shelf customer support. Plus a quick shout-out to the guys at Precise Bits — you really know your stuff!



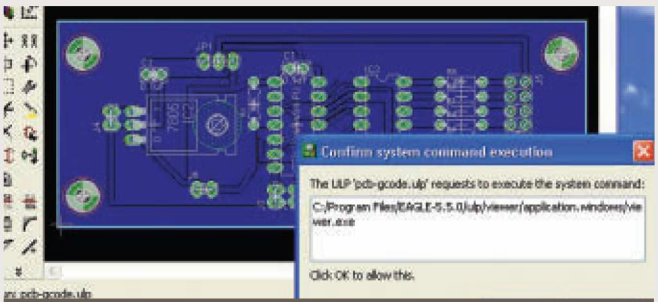
■ FIGURE 11. Type "RUN" in the command line on EAGLE to start PCB-GCODE.



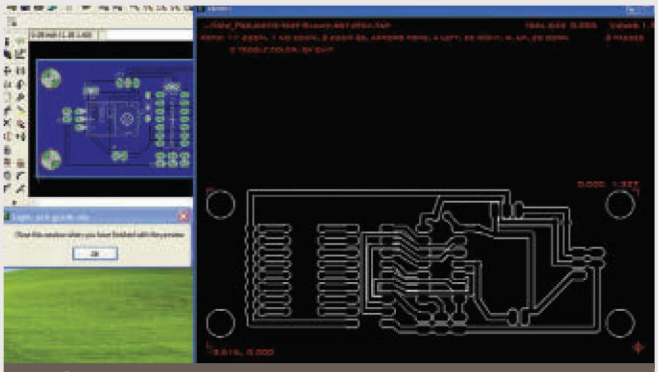
■ FIGURE 12. Choose the "Setup" file.



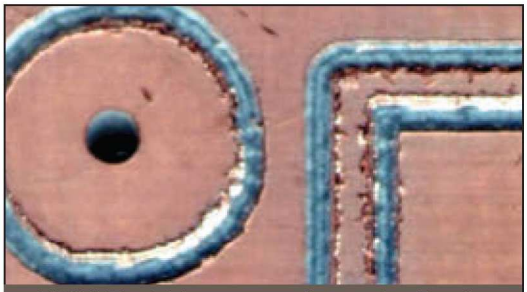
■ FIGURE 13. The Generation Options tab from PCB-GCODE.



■ FIGURE 14. PCB-GCODE prompts you to open the Windows Viewer to see the results.



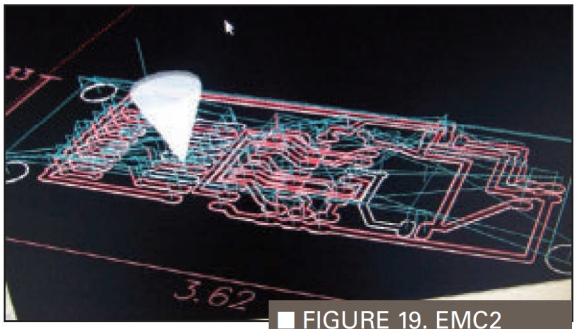
■ FIGURE 15. The trace isolation "etch" path generated by PCB-GCODE.



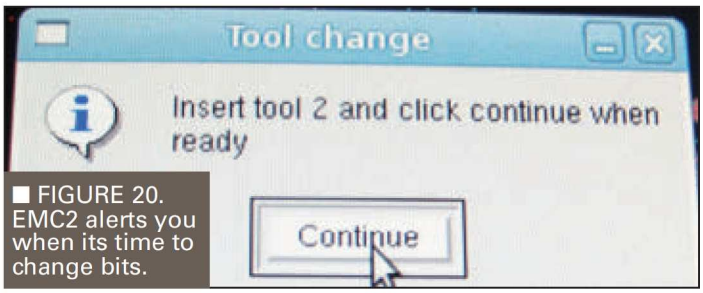
■ FIGURE 16. Test cut #1 shows the Z axis depth was too deep resulting in burred edges to the copper traces.

■ FIGURE 17. Test cut #2 shows the Z axis at the correct depth. Note the underlying PC board material is mostly intact.

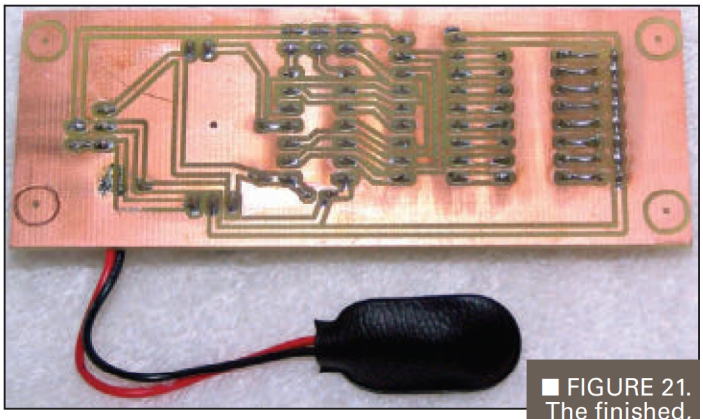
■ FIGURE 18. Copper clad PCB material ready to be cut.



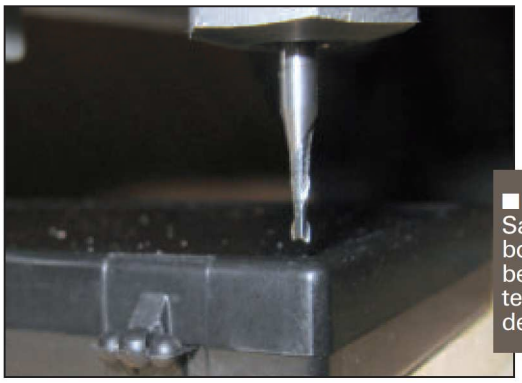
■ FIGURE 19. EMC2 running the etch pass on the v90.



■ FIGURE 20. EMC2 alerts you when its time to change bits.



■ FIGURE 21. The finished, working trace isolation version of Das Blinken Board!



■ FIGURE 22. Sacrificial plastic box ready to be test-cut for testing the bezel design.

the z axis off just enough to release the paper.

It's common to set the z axis a bit high since we can always go back and etch deeper a second time if needed. In the creation of the Das BlinkenBoard prototype, we actually did this "creeping down to copper" quite a few times to get it right. It's important to get this depth accurate because if you go too deep, the underlying board material is pulled up past the copper cladding resulting in a burred edge to the trace (**Figure 16**). When you get the depth set correctly, you will see evidence of the underlying board material being rubbed by the end of the bit, creating little circles in the path (**Figure 17**).

CUT IT OUT ALREADY

So, with our copper clad board mounted in place (**Figure 18**), it's just a matter of starting the run and keeping a close eye on the progress (**Figure 19**). (Note: Please use proper eye protection! The bit will throw debris!) Change the bits when asked (**Figure 20**) and then take your finished PCB and solder it up to see if it works (**Figure 21**). Ours did on the first try!

Now that we had success with the PCB creation, it was time to move on to cutting a case for Das BlinkenBoard. We used a similar process as that above, but since we didn't need to do the complex schematic and trace isolation work, we used VCarve Pro. We simply measured the various components and then layed out the holes/outlines for them. VCarve then created the GCODE and we used that same thumb drive to take it over to the v90 for cutting.

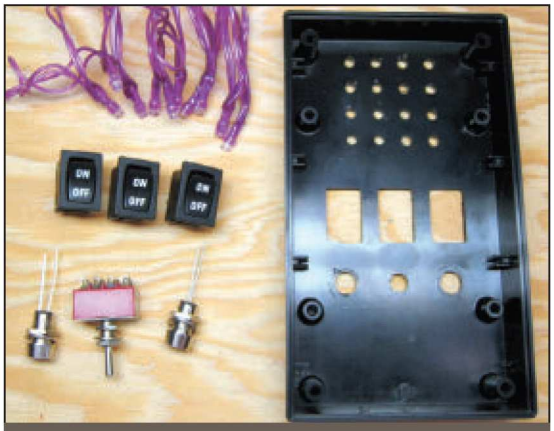
We placed a scrap piece of plastic in the CNC

DAS BLINKENBOARD UPDATE

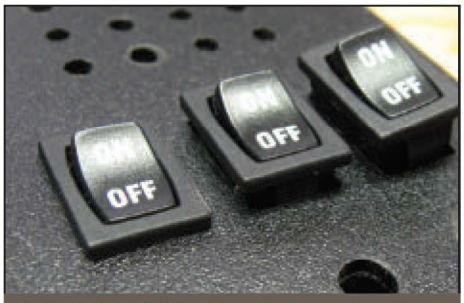
As I promised last month, some updated information has been posted on the Das BlinkenBoard website, including new software patterns focused on circular displays and a tutorial on how to reprogram the processor on the board using an inexpensive programming tool. Come by the website at **www.DasBlinkenBoard.com** and see what's new!

machine and did a quick test cut to be sure the orientation and fit were right (**Figure 22**), then placed a store-bought enclosure onto the CNC bed and cut out the component slots (**Figure 23**). After lining up all the components (**Figure 24**), we populated the holes beginning with the square snap-in switches (**Figure 25**), then the toggle switch, and lastly some LED holders (**Figure 26**). So now, Das BlinkenBoard has Das BlinkenBox (**Figure 27**)!

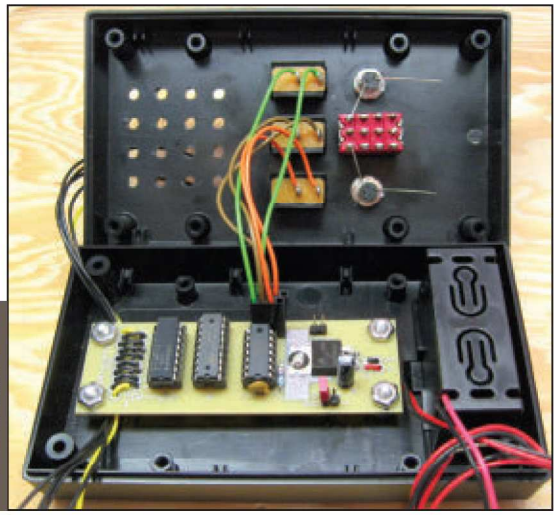
■ FIGURE 23. Finished front panel bezel for Das BlinkenBox.



■ FIGURE 24. Parts ready to go into the bezel.



■ FIGURE 25. The snap-ins switches shown here in various states of insertion.



■ FIGURE 26. All controls and most of the LEDs mounted in the bezel.

CUT! THAT'S A WRAP!

As you can see, besides cutting CD-ROM earrings it's quite possible to create your own professional-grade circuit boards and front panel bezels right in your own shop using a CNC machine. If you've been looking for an excuse to get into CNC systems, the time and money you'll save over mail-order PCB services, added to the frustration you'll save yourself over hand-fabricating your electronic enclosures, makes a CNC system pretty tempting! There are plenty of folks already involved in CNC and quite a few of them are ready to help out the new folks just starting out.

If you want to see some impressive things created by CNC owners, go by the PROBOTIX online forums and check out the pictures. As always, if you have any questions, feel free to drop me an email at vern@txis.com. **NV**

■ FIGURE 27. Das BlinkenBox front bezel with parts mounted.