

Basic Electronics

#4A

By Ron C. Johnson

Get out your drill, pliers and soldering iron. This time around we're building the speed control circuit I showed you in last month's segment. Actually you'll need the drill for more than putting holes in the chassis and printed circuit board because, when you're done here you will have a speed control which will turn that single speed drill into a variable speed model.

The accompanying photograph shows my version of the speed control in a small aluminum box about 6 cm by 8 cm by 13 cm. You just plug the drill into the AC socket on the top and plug the AC power cord from the box into the wall outlet. In this article we'll take the schematic of the speed control, convert it into a printed circuit board layout and assemble the parts into a working prototype. In the process, we'll look at several ways to make printed circuit boards: how to generate the artwork, and how to eventually get the pattern you designed onto a copper clad board.

Before we start into any part of this project an important warning must be made: **BE CAREFUL...** This project involves building and testing a circuit which connects directly to the 115 volt AC power line. **THIS IS A DANGEROUS VOLTAGE!** Disconnect the circuit from the AC power before soldering or touching the circuit board or components. If you must test the board while it is powered up make sure that no part of your body comes in contact with the circuit. A good rule of thumb is to test with one hand in your

pocket and the other holding the test lead. If you have an oscilloscope it should be isolated from ground by a isolation transformer or proper dif-

In last month's segment we quickly ran through the operation of this circuit. A little later on we'll expand on that a bit. You'll also notice, elsewhere in this

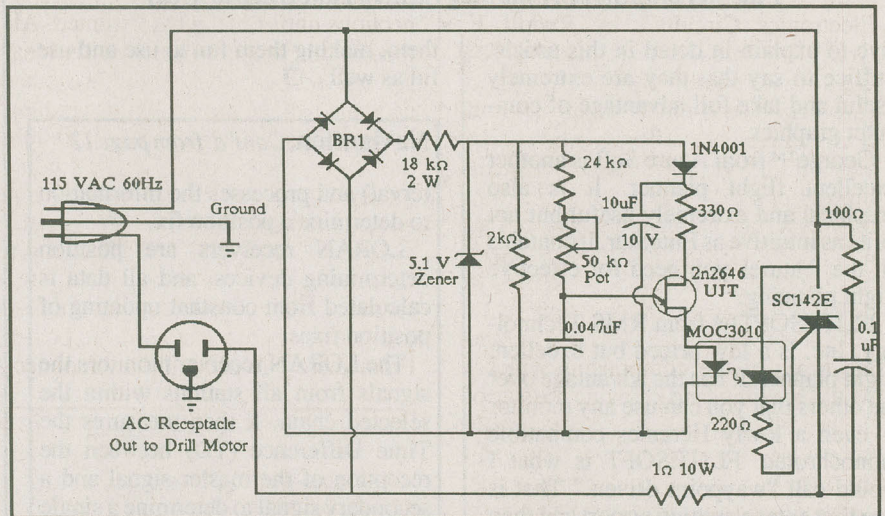


Figure 1. Variable Motor Speed Control Schematic with Corrections

ferential measurement techniques must be used. **DON'T TAKE ANY CHANCES!** (I don't want to hear about any readers going up in smoke... It's bad for the magazine's circulation, not to mention yours!)

First order of business: I must apologize for an error in the schematic diagram of the motor speed control shown in the last segment. The connections to the bridge rectifier were shown incorrectly. Figure 1 shows the correct version in addition to a couple of other changes in circuit components which I found necessary (or expedient) when I constructed the circuit.

article, a sidebar where I discuss the steps you should follow in the development of any project.

Okay, the construction details of this project:

Construction

Ideally, a speed control for an electric drill should be built into the drill itself, as it is in the commercially available variable speed drills. However, I didn't think this was feasible, and even if it was possible to fit the circuit inside your drill it probably would not be safe. The next best situation was to have a

small box, (the one I used was larger than necessary), with a speed control adjustment mounted on it, in line with the drill. After finding a suitable aluminum box at Radio Shack I knew what the maximum dimensions of my circuit board should be. I mounted the speed control potentiometer into the top, the AC line cord into one end and an AC outlet (chassis type) into the other end. Having done all this I could see what size to make the board and how I should position it to avoid any problems fitting the external wiring into the box.

From this point I could start the process of converting the schematic into a working prototype. As always, the first step was to breadboard the circuit in order to test it and work out any 'bugs'.

I should confess that this circuit is not my own design, but one I found in a book called "The Encyclopedia of Electronics Circuits", by Rudolf F. Graf. This book is published by Tab Books and contains over 1300 circuits categorized into ninety-eight sections. It is relatively inexpensive and can be very useful for finding circuit ideas and innovations. Anyway, the circuit from this book, (as usual), required a couple of initial modifications to get it working on the breadboard. The next step was to design a printed circuit board layout.

I knew the physical sizes and mounting types of the components which I would be using, and knowing the size of the box all of this would go into I could lay out my circuit board. One way to do this is to lay out the actual components, or paper cut outs of the components, onto a piece of paper the size of the final board, rearranging them until you obtain a satisfactory design. Next you draw in the locations of the pads which the component leads will be soldered to and interconnect them. Use pencil because it will probably take a few attempts and rearrangements before you get it right. With a board of this complexity you won't need to use jumpers (to jump over tracks to make connection to another part of the board); but this can happen with more complex boards.

In my case I used a drawing program on my computer (PC Paintbrush IV) to do the same thing. First, I made up drawings of each component I would use, and then, using the gadget box I

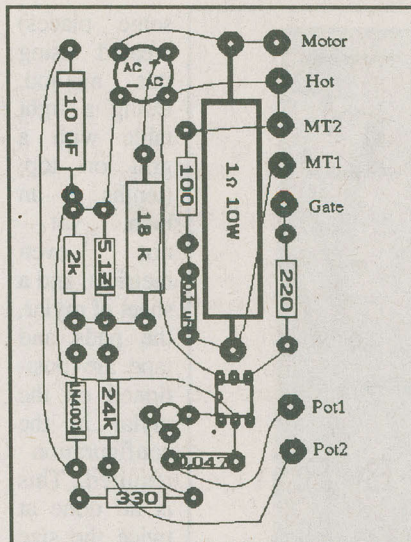


Figure 2. Component Side of PC

moved them around into the positions I wanted. (See Figure 2). Next I drew in pads and experimented with interconnections until I had what I wanted. Although this program is not really ideal for this sort of work (there are other dedicated PC board design programs), it wasn't too difficult to adapt it for this purpose. I then took this initial component layout and got rid of the components themselves while leaving the pads and lines in place. Next I used a function of the program to flip the layout so that I was looking at the bottom view, or solder side of the board. From there I re-drew the interconnec-

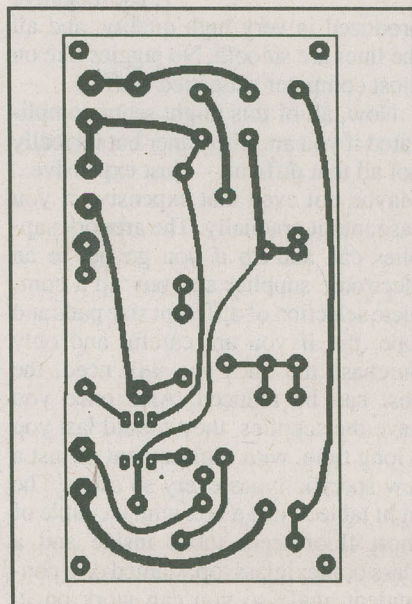


Figure 3. Solder Side of PC Board

tions as wider traces (Figure 3) and finally inverted the whole thing so it looked like a negative (Figure 4).

This gave me a design for the board layout but, as we'll see shortly, I took a different route to actually generate the printed circuit board.

Circuits As Art

Although this circuit is not extremely complex I wanted to make a circuit board photographically rather than using the crude resist pen method to draw the traces on the copper clad board. There are several ways to make PC boards. If you are using a negative

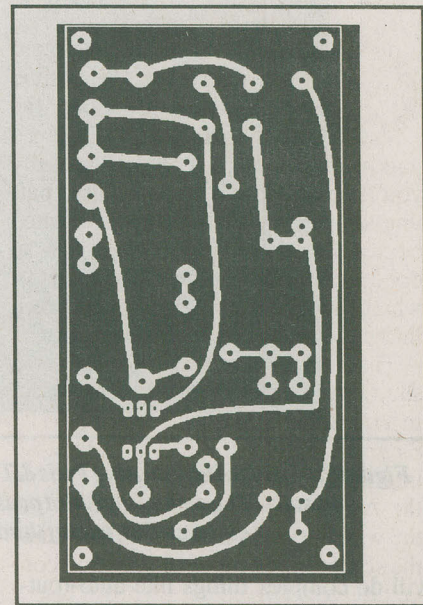


Figure 4. Negative of Printed Circuit

photoresist process you must generate a photographic negative in which the copper traces are transparent. If you are using a positive type process the negative must have everything *but* the copper traces transparent. Either way, you have to generate a high quality negative of the solder side layout which can be used in the photographic board etching process. First let's talk about various ways to get the artwork used in making a negative.

As I mentioned, the ideal method for generating artwork for PC boards these days is, not surprisingly, with a computer. There are all kinds of specialty software packages, such as Tango or Orcad, which not only allow you to lay out the pads and traces on screen, but

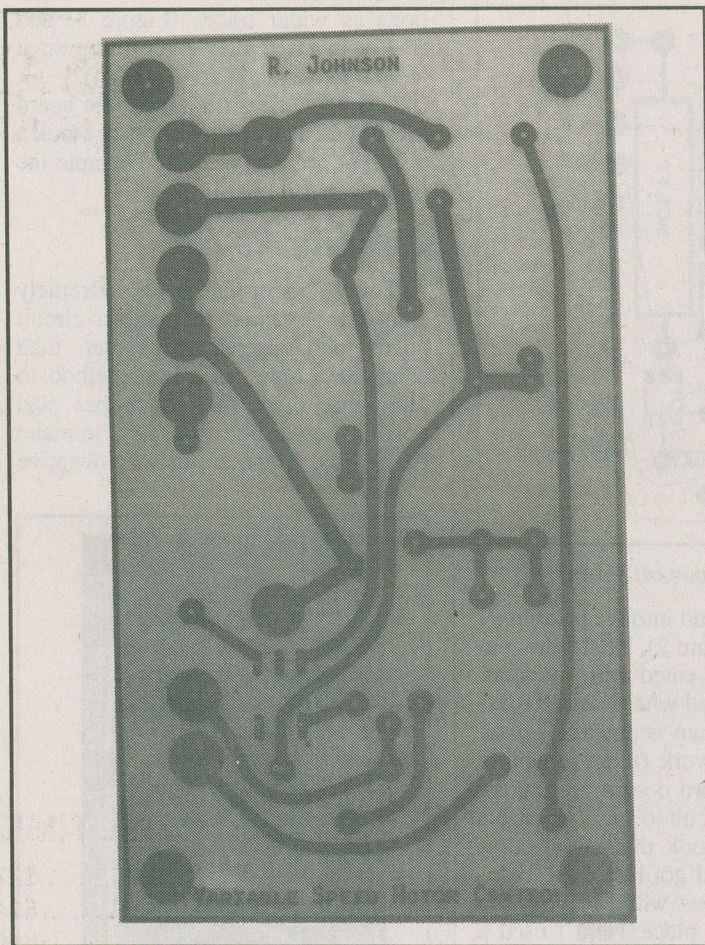


Figure 5. PC Board layout using Pads & Tape. (This layout contains a mistake. The second and third pads at top left should be connected instead of third & fourth).

will do complex things like auto-routing of traces, generation of lists of components, and lots of other goodies. Once you have the layout in memory you can print it out, either using a plotter, or a laser printer, which can then be used to make the negative. Although I had a layout, produced with my drawing program, I decided to use a different method to produce my negative for two reasons: First, I encountered some difficulties in getting a high quality, correctly sized print out of the Paintbrush drawing I created. Second, I wanted to show you how it could be done without needing a computer.

As for those computer circuit design programs... a future article, perhaps...

So how did I do it?

I went back to the old, tried and true method using double size tape and pads. For years, professionally printed circuit boards were (and probably still are in

some places) created using this method. Using a light table with a grid on top, (tenths of an inch, yet — not even metric!), and a sheet of mylar, the pads and tape are positioned on the mylar in the configuration required. This is all done at twice the size the board will eventually be (sometimes at four times the size for complex boards). When the artwork is finished, it is taken to a specialty photographic shop where they produce a negative of the artwork at half its original size. (The negative


produced is very high quality, and all the lines are smooth. No jaggies like on most computer generated stuff...)

Now, all of this might seem complicated if you are a beginner but its really not all that difficult — just expensive... Maybe not even that expensive if you ease into it gradually. The artwork supplies can add up if you go out to an electronic supplier and buy up a complete selection of different size pads and tape, but if you are careful and only purchase the ones you will need, the cost can be reduced. Also once you have the supplies, they should last you a long time, with replacement of just a few specific types every so often. The light table is just a box with a couple of short fluorescent tubes inside and a glass or plexiglass top, slanted on a convenient angle so you can work on it. You can easily build one if you decide

to go this route. (All in all, though, you can see why the computer generated artwork approach is better if you have hardware already. Software may cost you anywhere from one hundred dollars, for something basic, up to several thousand, for the high tech programs, but once you have it you can do a lot with it.)

As for the photographic reduction... just look in the yellow pages under graphics or photographics and phone around. You will eventually find someone who can do the job. For a negative the size of the board in this project it cost me about five bucks, (plus the GST, of course, which some people say stands for... no, I won't repeat it), but I got a deal. It might cost you anywhere up to twenty from a commercial shop.

So I laid out my artwork, (see Figure 5), had a negative made and then used the negative to produce a board. (Figure 6) I should mention here that (blush) I



PROTEL
Professional
Quality CAD
Software.

Guaranteed Lowest Prices on Protel Products.

Schematic (DOS)	\$ 549
Design Works (Mac)	\$1099
Easytrax (DOS & Mac)	\$ 449

PCB layout with through hole technology, DRC, pad-to-pad routing, postscript printing.


Autotrax
Non-Extended (DOS) **\$ 999**
(Mac) \$1099

PCB layout with SMT, netlist import, auto load, manual route, DRC.

Autotrax
Extended (DOS) \$1449
(Mac) **\$1699**

As above including full Auto Place and Autorouter.

Combinations of packages offer savings from 5- 20%. Free full-function Demonstration Diskettes. 30 day money back guarantee. Free and knowledgeable Technical Support is provided as always by the experts on our in-house PCB Design Staff.



An Engineering Support Company

Innovative Devices, Inc.
1119 Damelart Way, Brentwood Bay, BC
V0S 1A0 (604) 652-5240
Toll free AB & BC 1-800-661-2881

Circle No. 26 on Reader Service Card

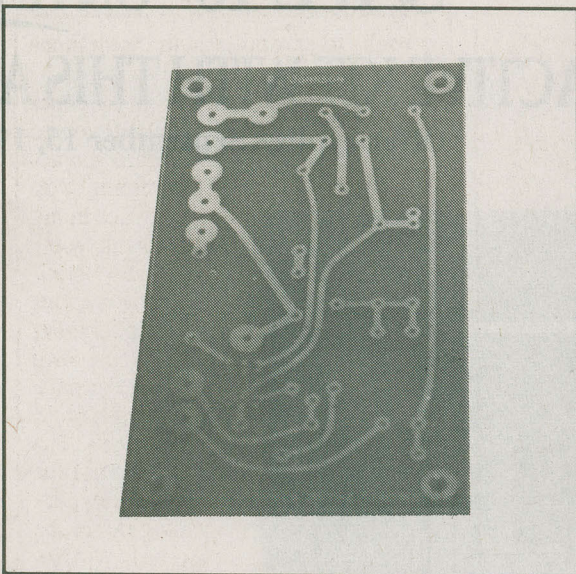


Figure 6. The Finished Printed Circuit Board

made a mistake on the artwork which I did not notice until I was mounting the components on the finished PC board (see the sidebar about prototypes not being the finished product). Normally I should have re-taped the artwork, had a new negative made up as well as a new PC board - but I didn't... The mistake was easily corrected by cutting one

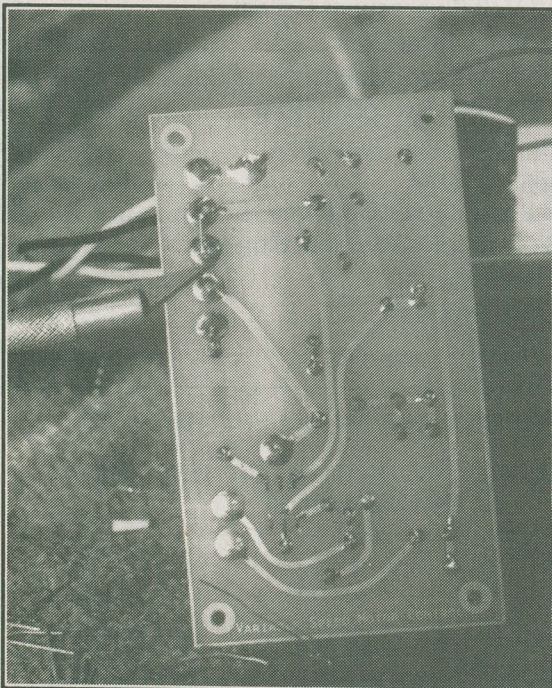


Figure 7. Solder Side of the PC Board with Components Soldered in. (The knife shows the track that was cut. Directly above, a jumper was soldered in.

track and soldering in a jumper on another, so that's what I did. (Figure 7) (By the way, Figure 3 shows the artwork with the correction made so you can use it without fear.)

The Board

As I mentioned I decided to use a negative, rather than positive, process to make my PC board. The negative process has been around longer, and it happened that I had access to the equipment to do it that way. As with artwork, though, if you do not have access to the resources to etch circuit boards, it can be expensive to get into.

In previous articles I have talked about using ferric chloride to etch boards that you have made using an etch resist pen. This stuff works fairly well, especially if you warm it up, but I should caution you here that you should be careful with it. If you happen to pour it down a drain it can eat right through the drain pipes. (I probably don't need to suggest how bad this is for the environment in general.) If you do use it, and dispose of it this way, make sure that it is well flushed out and diluted by a quantity of water. (It is soluble.) A better solution might be to find an alternate disposal method. Check with your local municipal government or an environmental group; they should know the details.

Back to the PC board...

It isn't the etchant that is the expensive or difficult part of getting from the negative to finished board. In order to use the negative to transfer the layout onto the copper clad board you must first sensitize the board to light. One way to do this is to use liquid photoresist. This is the old method where photoresist is poured onto the copper clad board and spread around into a thin layer using a

About Projects

Let's talk about the process involved in putting together this project. Even though the level of complexity of the circuit is relatively simple, the design procedure is similar. To some extent we have followed this procedure in the simpler circuits constructed in this series, but let's look at it again.

If your main desire is to build a motor speed control and you use the schematic, component values, and circuit board layout shown here you will probably end up with a project that works. But, let's be honest here: how many times does a circuit you find in a magazine, or elsewhere, work perfectly when you power it up after building it. In my experience: less than ten percent of the time. Why? The reasons are many and varied: errors in the schematic (like I gave you last month), variations in components and tolerances, differences in construction techniques, errors in construction, faulty components, etcetera...

But then, it wouldn't be any fun if everything worked perfectly the first time...

So what about this process? In electronics, as in other scientific and engineering oriented fields, projects are often organized in phases. The first phase, after deciding what it is you want to build, is to decide what your specific needs or requirements are. For example, in the motor speed control we ask ourselves: "What kind of a motor do we want to control? Do we want to control all the way from zero RPM to maximum RPM? What are the electrical requirements of drill we will control? Physically, how large and what shape should the speed control be? What are the cost considerations? What about parts availability?" The list could go on, and some of your answers may be different than mine.

After deciding on your requirements you go to the design phase. In this case you may just capitalize on the work I have done in designing the prototype, or, you may want make the circuit physically smaller, or able to drive a larger motor. Often, during the design phase, you will go

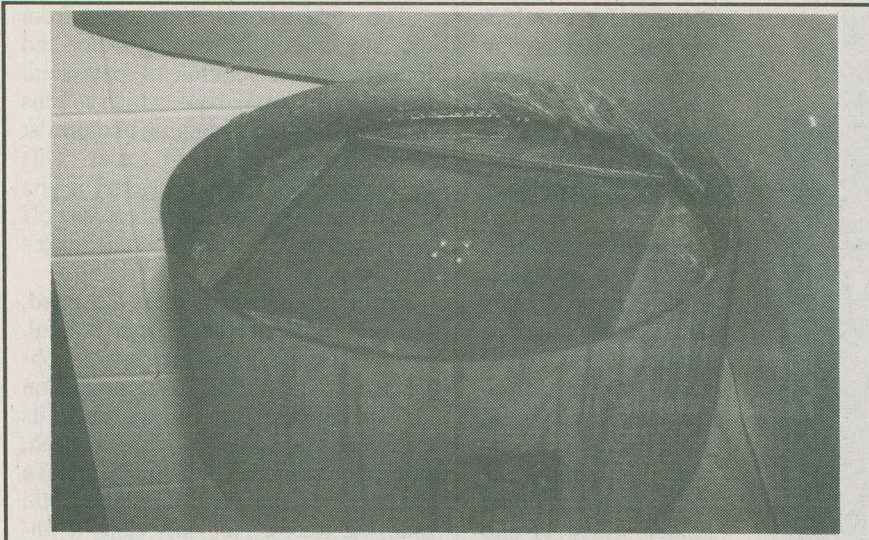


Figure 8. A Spinner Used to Spread Photoresist

Photo courtesy of Northern Alberta Institute of Technology (NAIT) Electronics.

centrifugal spinner. (Incidentally, before any kind of photoresist can be applied, the copper clad board must be cleaned of all oxidization using steel wool or sand and a scouring pad.) Figure 8 shows one of these spinners with a build-up of photoresist around the edge. The photoresist itself is quite toxic, requiring careful handling and good ventilation. A spray-on type is also available, with the same drawbacks. Also, both types are expensive. Another way to go is to buy pre-sensitized boards wrapped in black, opaque plastic which avoids the need to handle chemicals. These too, can be expensive. It should be noted that all of these sup-

plies are available from electronic suppliers such as Cardinal Electronics or Electrosonic.

Another way, newer, and the way I chose to do it, is called a dry laminate process. In this process a special, dry film is laminated onto a clean, de-oxidized copper clad board using a special laminator. The dry laminate is light sensitive, similar to photoresist but is easier and safer to handle. The procedure for developing is slightly different than for photoresist and uses safer chemicals as well. With dry laminate photoresist, as with the liquid, once the photoresist is on the board the areas which you want to remain on the board

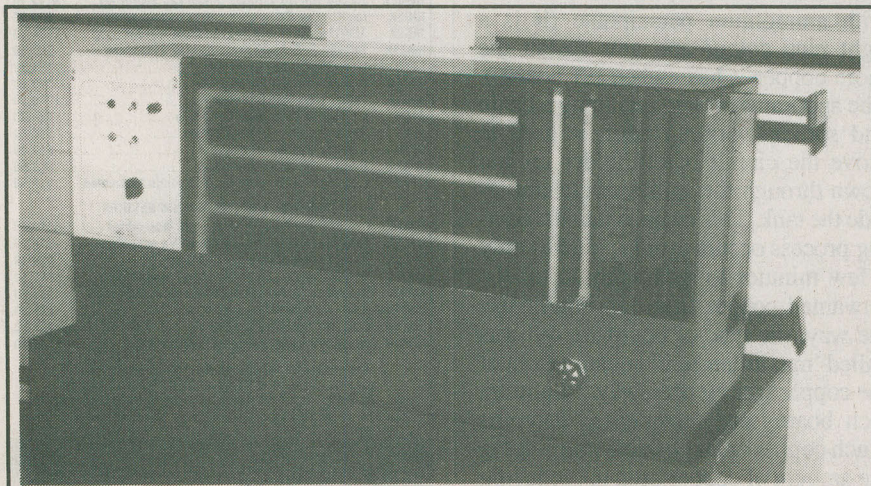


Figure 9. A Developing Unit

back to examine and sometimes change your requirements.

Once you have a design on paper it is highly recommended that you go to a breadboard stage before you actually try to build the circuit. You may have supreme confidence in my design - and I humbly thank you - but, still, I suggest that you breadboard the circuit first, for the reasons mentioned earlier. Breadboarding allows you to try the circuit with actual components and to make quick modifications, if necessary. When you are satisfied that the circuit will work, you can then use the same components in the prototype.

I used the word 'prototype' in the last paragraph purposely. After breadboarding the circuit, you will design (or use my design) and fabricate a printed circuit board to mount and interconnect the components as on the breadboard. We often hope that this first circuit board with operate satisfactorily, but unfortunately, all too often problem arise, even at this stage. That is why we call it a prototype. The circuit is a one of a kind, test run, which will prove whether or not the design will work in this final form. If it does, great! If not we may have to make further modifications to it. In the case of a home project you may be satisfied to make the modifications required by some sort of 'kluge' repair to the board: adding components or cutting tracks and adding wires. Of course, in a professional or manufacturing environment, the design would be reworked and subsequent prototypes would be constructed until a finished product was obtained. (The truth is that I've worked in manufacturing and it's surprising how often products come out with post-manufacturing fixes added to them.)

I've already alluded to the final step in the process which is that of testing. Once the prototype is build it should be tested against the criterion established back in the first step. The questions must be answered: Does this thing really do what I wanted it to do in the first place? Does it look right? Is it safe? Did the cost fall

see Projects, Cont'd. on page 23

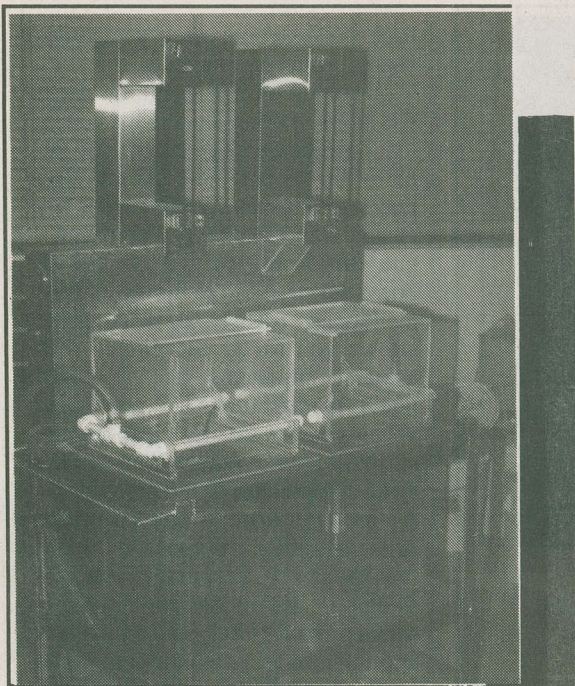


Figure 10. An Etching Tank Using Ammonium Persulfate.

as pads and traces must be exposed to a bright ultraviolet light. This is where your negative comes in: The negative is placed over the sensitized board and exposed to the ultraviolet light. Special UV fluorescents can be used, and I have heard that even direct sunlight will do the job if the board is exposed long enough.

The light I used was a xenon type light in a special flip-top exposure table. This unit uses vacuum to hold the negative firmly in place against the PC board. First the board and negative are placed on the table and the glass is lowered over them. Next, the vacuum is turned on, which pulls the negative down hard against the PC board. The whole top of the unit pivots and is turned over so that the board and negative face down at the light which is then turned on. A timer shuts the light off in about three minutes.

Now that the board has been exposed it must be developed like a photograph. Developing the board basically hardens the photoresist in the areas where it was exposed to light making it impervious to the etchant used later. With the old type of photoresist the chemicals used for developing were pretty strong. Another advantage of the dry laminate process is that it uses sodium carbonate

solution for developing. This is a relatively harmless chemical and the results of the developing process are at least as good as with the older process.

Figure 9 shows a developing machine. Basically this is just a tank that is plumbed with pipes which have spray nozzles set to spray the entire board with developer. In this case the board is clamped into a sliding holder which is slid into the developer from the side. A separate chamber is used to wash the developer off the board with water before removing it from the machine.

After the board is developed it is ready to be etched. The etchant used in this particular system was ammonium persulfate. We talked about the disadvantages of ferric chloride before. Ammonium persulfate has some real advantages: Although it is also heated to improve its speed of etching it will etch at room temperature (albeit slowly). It is also more environmentally friendly and doesn't attack metals quite as violently as ferric chloride. The etching tank shown in Figure 10 is a dual unit. You will notice the tank on the left filled with ammonium persulfate (it is a light blue colour and gets darker the more copper it has etched off boards). The apparatus on top, including a chain and sprocket arrangement, is used to move the circuit board holder up and down through the sprays of etchant inside the tank. This ensures that the etching process occurs evenly. It takes only a few minutes to etch away all of the unwanted copper from the board. (By the way, the type of board we did was called 'maximum etch' because most of the copper was etched away. Minimum etch boards are designed to leave as much copper on as possible minimizing the amount of copper used up, reducing etching time and prolonging the life of the etchant.)

The etched board is removed from the tank, rinsed in running water and immersed in a solution of potassium hydroxide, a strong base, which softens and dissolves the remaining photoresist on the copper pads and traces. This chemical is not really necessary if you don't mind scouring the photoresist off with steel wool or sand but it makes things easier.

At this point you could use the board, but one further refinement is helpful. From electronic suppliers, you can obtain a chemical called 'Cold Immersion Tin Plating Solution'. I'm not sure of its chemical make-up but when a clean, printed circuit board is immersed in it a chemical reaction deposits a layer of tin onto the traces. Not only does it improve the appearance of the board but it inhibits any oxidization of the copper which would make it difficult to solder to later.

AMAZING SCIENTIFIC & ELECTRONIC PRODUCTS

LASERS AND SCIENTIFIC DEVICES

VRL2K	3mw Vis Red Laser Diode System Kit\$159.50
LLIS1K	Laser Beam "Bounce" Listener Kit\$199.50
LHC2K	Visible Simulated 3 Color Laser Kit\$44.50
LC7	40 Watt Burning Cutting Laser Plans\$20.00
RUB4	Hi Powered Pulsed Drilling Laser Plans\$20.00
LGU40	1 to 2mw HeNe Vis Red Laser Gun Assembled\$199.00
LLS1	Laser Lite Show - 3 Methods Plans\$20.00
SD5K	See in the Dark Kit\$299.50
EML1K	Electromagnetic Coil Gun Kit\$69.50
MCP1	Hi Velocity Coil Gun Plans\$15.00
LEV1	Levitating Device Plans\$10.00
EH1	Electronic Hypnotism Techniques Plans\$10.00

HIGH VOLTAGE AND PLASMA DISPLAY DEVICES

HVM7K	75,000 Volt DC Variable Output Lab Source Kit\$149.50
IOG3K	Ion Ray Gun Kit, project energy without wires\$69.50
NIG9K	12V/115 VAC Hi Out Neg Ion Generator Kit\$34.50
EMA1K	Telekinetic Enhancer/Electric Man Assembled\$99.50
LG5K	Lightning Display Globe Kit\$54.50
BTC1K	Worlds Smallest Tesla Coil Kit\$49.50
BTC3K	250KV Table Top Tesla Coil Kit\$249.50
BTC5	1.5 Million Volts Tesla Coil Plans\$20.00
JL3	Jacobs Ladder - 3 Models Plans\$15.00
GRA1	Anti Gravity Generator Plans\$10.00
PFS20	Plasma Fire Saber Assembled\$69.50
DPL20	Dancing Plasma to Music and Sounds Assembled\$79.50

SECURITY AND PROTECTION DEVICES

ITM10	100,000 Volt Intimidator up to 20' Assembled\$129.50
IPG70	Invisible Pain Field Blast Wave Gun Assembled\$74.50
PSP4K	Phasor Sonic Blast Wave Pistol Kit\$59.50
LIST10	Infinity Xmit, Listen in Via Phone Assembled\$199.50
TAT30	Automatic Tel Recording Device Assembled\$24.50
VWPM7K	3 Mi. FM Auto Tel Transmitter Kit\$49.50
FMV1K	3 Mi. FM Voice Transmitter Kit\$39.50
HOD1K	Homing/Tracking Beeper Transmitter Kit\$49.50

EASY ORDERING PROCEDURE TOLL FREE 1-800-221-1705
or 24 HRS ON 1-603-673-4730 or FAX IT TO 1-603-672-5406
VISA, MC, CHECK, MO IN US FUNDS. INCLUDE 10% SHIPPING. ORDERS
\$100.00 & UP ONLY ADD \$10.00. CATALOG \$1.00 OR FREE WITH ORDER.

INFORMATION UNLIMITED
P.O. BOX 716, DEPT. ET2, AMHERST, NH 03031

Circle No. 28 on Reader Service Card

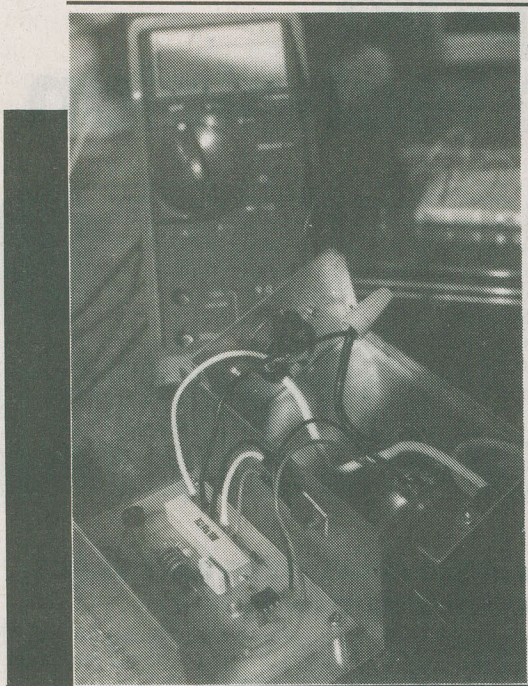


Fig. 11. The Completed Project

After a couple of minutes of tinning the PC board can be trimmed and drilled. Drill bits for electronic component leads are very small and must be purchased at electronic suppliers or specialty tool suppliers. A Dremel type tool, especially with a press attachment, is useful here as alignment can be difficult and tiring after a few dozen holes.

Well, there we have the PC board manufacturing process. Hopefully, it was of some interest to you. I suspect, however, that considering the costs, and the amount you might use it, you may not rush right out and set up your own circuit board manufacturing facility. However, as a hobbyist you will still want to make PC boards. Let me suggest a couple of things: try to find an individual or company who has all this stuff and will let you use it from time to time. Often, a friend who has access to this kind of equipment could get the use of it on weekends or evening at little or no cost to you. Alternately, find a local manufacturer of PC boards who will make the board from the negative. I found one small company, operated by one person out of his basement, who would make up one board at a time, trim, drill, plate, etcetera for fairly reasonable prices. If you do decide to set up your own facility, start small, and above all be careful. Make sure that

ventilation is good and storage is adequate to keep chemicals safely out of reach of children and those not familiar with their use.

Let's continue and look at how the printed circuit board is used in the rest of the project.

Stuffing The Board

As I mentioned, I made a mistake on the layout of the board. When I finally noticed this I used a scalpel type knife to cut the unwanted track and then soldered in a jumper in the right place. Although not too cool, it was effective and cheaper than starting over. (Who says I'm tight?) Next I installed the components onto the board, forming

the leads with needle-nose pliers. I mention this because it is a frustrating way to form leads, usually resulting in oddly shaped leads and a generally messy board. There are a couple of lead bending tools on the market which, though made out of cheap plastic, work very well. But don't be fooled by the material they are made of: they will still cost you more than a family outing at the Golden Arches.

Circuit Operation

Referring back to the schematic in Figure 1, a couple of things should be pointed out. First, the power triac: The original number of the part used in last month's article was a 2N6347, a standard 15 Amp triac. I replaced it with an SC142E, which is a 10 Amp version (more than enough for this application) in a TO-220 package

Projects, Cont'd. from page 21

within the budget? (Never seems to in my case...)

Well, this stuff is just common sense, but it might help to ease some of the frustration when things go wrong, to have a plan of how you expect to get a project done. Remember Murphy's Law: Anything that can go wrong, will. But, hey, that's where the fun is, right? □

with an *isolated tab*. This is important. Although it is possible, even probable in retrospect, that the triac need not be chassis mounted for heat dissipation at all, I did it anyway. But I was concerned that, unless very good quality insulation was obtained using insulating hardware, a shock hazard might exist. To avoid any problem I chose a triac with an isolated tab so that no circuit voltages would be present on the tab which is bolted to the chassis. I also made sure that the third prong ground wire from the AC line cord was securely connected to the chassis. This ensures that any voltage contacting the chassis would be diverted to ground.

Figure 11 shows the circuit board mounted in the box on short standoffs and the wiring to the power triac, potentiometer and AC cord and outlet.

Another difference between the circuit I built and the original schematic involved the timing components of the relaxation oscillator used to fire the triac. Even though the circuit worked well in the breadboard stage, I found

see Basic, page 26

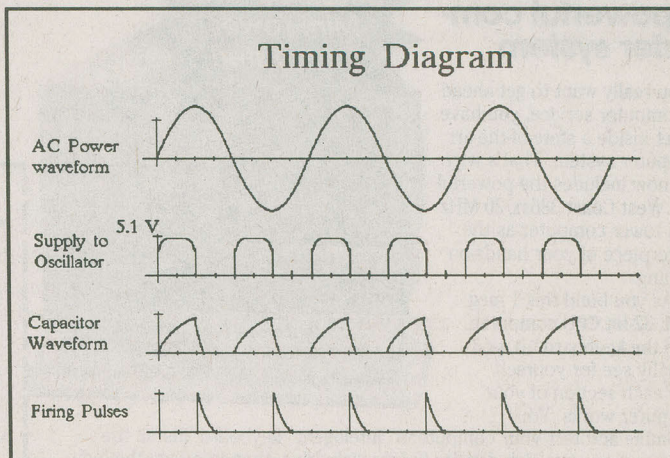


Figure 12. Phase Control Waveforms

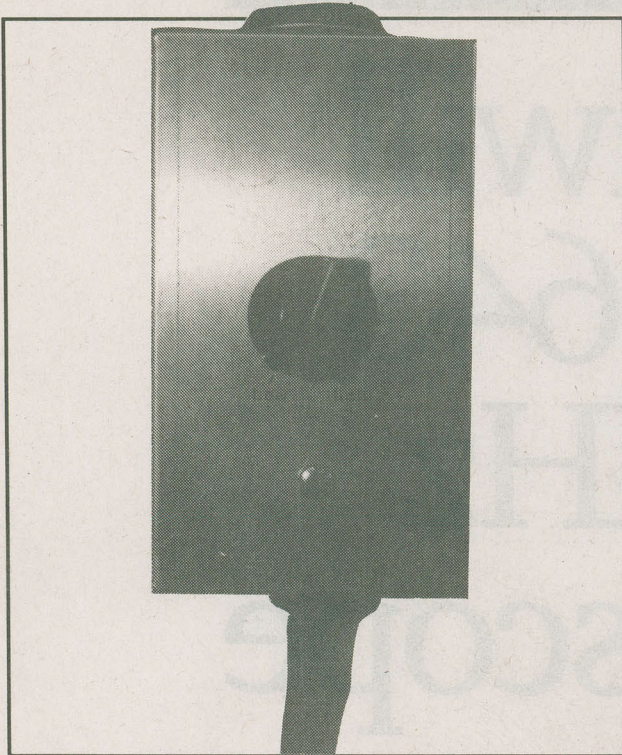


Figure 13. The Completed Variable Motor Speed Control

Basic, cont'd. from page 23

that the firing time was too long in the prototype. This caused the circuit to fire erratically at low speeds because the capacitor would charge up part way on each half cycle, finally reaching the firing point at unpredictable times, turning on the triac for a short burst of speed. The effect was that, as the potentiometer was turned counter-clockwise (increase of resistance), after a certain point the drill would periodically surge to a higher speed. I used my oscilloscope to check the timing of the capacitor charge/firing/discharge curve and found that it was too slow. After experimenting I found that a slightly smaller value of capacitance (0.047 μ F instead of 0.1 μ F) and a resistance of 24 k ohms in series with the 50 k ohm potentiometer gave a satisfactory control range from very slow to, almost, full speed output.

One final comment about the operation of the circuit: In last month's segment I included some timing diagrams which showed how the relaxation oscillator was used to fire the power triac through the opto-triac. These were correct as far as they went but if you end up troubleshooting this circuit with an os-

illoscope you will see waveforms which look slightly different, especially with respect to the full-wave supply to the oscillator. Figure 12 shows the same timing diagram as before but includes the actual supply waveform, as seen across the 5.1 volt zener diode. You will note that whenever the UJT fires, the supply voltage drops to zero for the rest of that cycle. This is because, as soon as the UJT fires, which turns on the triac, there is no longer any voltage impressed across the UJT circuit. The power triac is connected directly across the UJT circuit and when it turns on it becomes a short—a

closed switch—which has practically no voltage dropped across it. Until the cycle ends and the triac turns off, the voltage across the firing circuit is zero. After the end of the cycle the triac turns off and the whole thing starts over again.

The important point here is that, when testing with an oscilloscope, the supply voltage waveform (measured across the 5.1 volt zener diode) will look like a pulse which gets narrower as the speed of the drill is increased.

Conclusion

In spite of the 'chasing around' required to get set up to do photographic printed circuit boards, this is not a difficult project and the finished product worked with a minimum of after-construction troubleshooting required. If you try to build it you may find that an oscilloscope would be useful to determine exactly what is going on in the timing and firing circuits. In this article I purposely described the details—and options—of many of the steps so that those of you who are new to building projects might gain some perspective on the process. Hopefully this will help you to get started...

Next time... How about some op amps and comparators? See you then. □

Earn Your B.S. Degree in ELECTRONICS or COMPUTERS



By Studying at Home

Grantham College of Engineering, now in our 41st year, is highly experienced in "distance education"—teaching by correspondence—through printed materials, computer materials, fax, and phone.

No commuting to class. Study at your own pace, while continuing on your present job. Learn from easy-to-understand but complete and thorough lesson materials, with additional help from our instructors.

Our Computer B.S. Degree Program includes courses in BASIC, PASCAL and C languages—as well as Assembly Language, MS DOŠ, CADD, Robotics, and much more.

Our Electronics B.S. Degree Program includes courses in Solid-State Circuit Analysis and Design, Control Systems, Analog/Digital Communications, Microwave Engr., and much more.

An important part of being prepared to *move up* is holding the right college degree, and the absolutely necessary part is knowing your field. Grantham can help you both ways—to learn more and to earn your degree in the process.

Write or phone for our free catalog. Toll free, 1-800-955-2527, or see mailing address below.

Accredited by the Accrediting Commission of the National Home Study Council

GRANTHAM College of Engineering Grantham College Road Slidell, LA 70460 USA