FEATURE

# Basic Electronics #3a

by Ron C. Johnson

Welcome back to the basics! Those of you who have been following this series know that this is 'boot camp' for semiconductor electronics — the place where you can get 'up to speed' on active components and circuits. If you are new to the series, don't worry: you can jump in here anywhere and find something useful and interesting to learn or to do.

Our last segment did a fast fly-past on transistors from the theoretical through applications. We looked at biasing, amplifiers, oscillators and some switching circuits. This month we'll build a couple of those circuits so that (if you are new to this) you can see what it takes to build a circuit from scratch. Hopefully you all have a DC power supply and a multimeter, (Did you build the simple power supply from the February issue?), and some of the tools needed to breadboard and build the circuits.

First, let's take another look at the phase shift oscillator from last month's article. (See Figure 1.) This is an easy project to build, (not so easy to design) which is inexpensive and useful as an audio signal source for testing circuits on your bench. While it will not replace a function generator it allows you to build something easy to start with. (Maybe we'll build a full fledged function generator later on.) Let's approach the subject from the perspective of how and why it was designed the way it was and then look at some construction details.

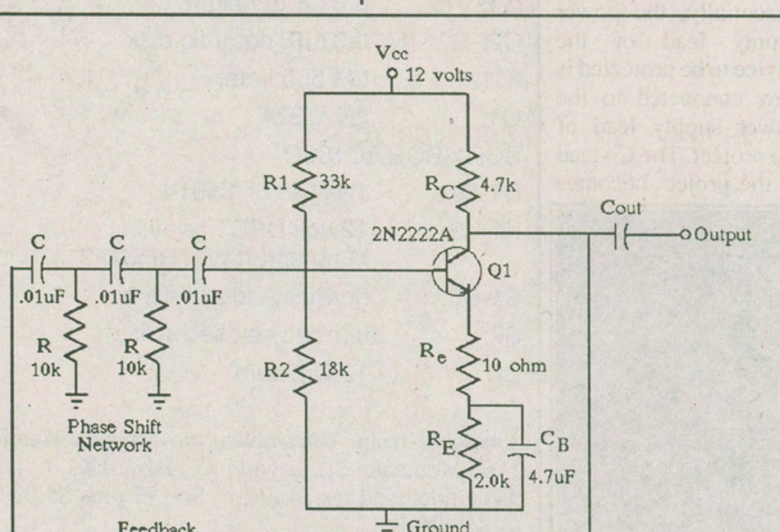


Figure 1. A Phase Shift Oscillator

What's in this oscillator? Let's review what we said last month. First, we saw how we could design a simple transistor amplifier stage using one transistor, a couple of capacitors, and some resistors. The circuit was set up to amplify an AC signal (for our purposes, in the audio range). The next step was to meet the criteria for oscillation which was to feed back the output signal of the amp to the input, in phase, with an overall gain of one. The amplifier inverts the signal, (180° shift), so the feedback network must invert it again to achieve the total 360° phase shift. The phase shift network, (three resistors and two capacitors) provides the phase shift at one particular frequency, but it also adds an attenuation of 1/29 so, to obtain an overall gain of one, the amplifier must have a gain of 29. (29 x 1/29 = 1) There is a formula for determining the frequency of oscillation of this circuit: $f = 1 / (\pi\sqrt{6RC})$. R is the value of both resistors and C is the value of all three capacitors in the phase shift network.

I wanted a frequency of oscillation of somewhere around 1 kHz but the exact frequency was not critical. Since there are two variables, (R and C), in the formula, some assumptions had to be made. I started by excluding capacitors above about 1 μF, especially electrolytics which usually have fairly high leakage and are bulky, both of which are undesirable in this situation. Also I ruled out very low values of capacitance because I knew it would result in very high values of resistance which I did not favour. I had some good quality .01 µF capacitors laying around so I decided to try them in the formula to start. Substituting in 1 kHz for the frequency and .01 µF for the capacitors I came up with a resistance of about 6.5 k ohms. It turned out that I was short on 6.8 k ohm resistors (the nearest EIA value) that day but I had lots of 10 k ohms so I decided to use them. Recalculating, using 10 k and .01 µF gave a frequency of oscillation of about 649 Hz which was okay for a simple audio oscillator.

The next step was to design an amplifier stage which will give a gain of 29. We didn't talk much about how to design a small signal transistor amplifier last month for good reason: it can get somewhat complicated even for a simple one. In addition, feeding the output back to the input, as we do in this oscillator, introduces some considerations that can make your head ache. We can't get into the whole story here but I'll give you enough to understand what the problems are.

The gain of a common emitter, voltage divider bias amplifier is determined by the ratio of the total collector resistance to the emitter resistance. (Gain = Rc/Re.) This is best proven using a page of mathematics showing how the emitter resistor controls the base current which controls the collector current, etcetera...a task we will avoid. However the design does centre around biasing the amplifier so that it will provide the gain of 29 even with its output connected back to its input through the phase shift network. This involves a series of calculations and approximations which start with the specifications (beta, for one), of the transistor you will use, DC operating point, AC operating point, the loading of the base voltage divider, input and output impedances, etcetera ad nauseam.

For those of you who already have a headache, you can skip this paragraph as I expand a bit on the design considerations. The DC operating point of the transistor is set by choosing the values for the voltage divider, R1 and R2. The DC operating point is the DC collector current and collector voltage of the transistor (which is the output) and sets the point around which any AC signal fluctuates. Collector current is controlled by base current and, in this case, base current is controlled by the voltage set by the values of the resistors, R1 and R2. Another consideration in choosing the values of R1 and R2 is that they influence the input impedance of the amplifier. When considered in parallel they have to be large enough to not load down the previous stage, which in this case is the output of the phase shift network, which is driven by the transistor. (Makes ya dizzy, doesn't it?) Getting back to the voltage set by these two: I said that this voltage sets the base current. Another problem to be designed away is that the resistance seen looking into the base of the transistor must be high enough to avoid loading down the voltage divider because that would cause our base voltage to turn out different than what we designed for. The resistance seen is determined by the two emitter resistors, in series, plus the bulk resistance of the transistor, $r_e$, all times the beta of the transistor. This same resistance is also in parallel with R1 and R2 and so influences the input impedance of the amplifier, as well. Here is where some of the problems come in: The collector resistance used in the gain formula is a parallel combination of the collector resistor and the impedance of the load connected to it. In this case the load connected is the phase shift network and the input of the amplifier. If the emitter resistor is changed to improve the gain or so that loading of the voltage divider does not occur, is also changes the input impedance which is part of the load, which changes Rc, which changes the gain. What this means is that whole procedure becomes something of a treadmill: every time a change is made anywhere in the circuit to change the gain, it has more far reaching affects because of the impedance changes it causes. You plough through the procedure several times to eventually get the gain required.

Wow! I'm glad that's over.

If it is possible to find this interesting, instead of just painful, it is that, designing even a simple circuit such as this, with transistors can be fairly complex. When I breadboarded this circuit, (after several tries at redesigning the gain stage) I finally got it to oscillate. However, because the gain was still a bit too high, the output was not a clean sine wave. It was distorted (very obviously not a clean sine wave shape when viewed on an oscilloscope). Distortion on a waveform is actually the presence of other frequencies, (usually harmonics of the fundamental), which can extend fairly high in frequency. (This became evident when the television in the next room came down with a case of

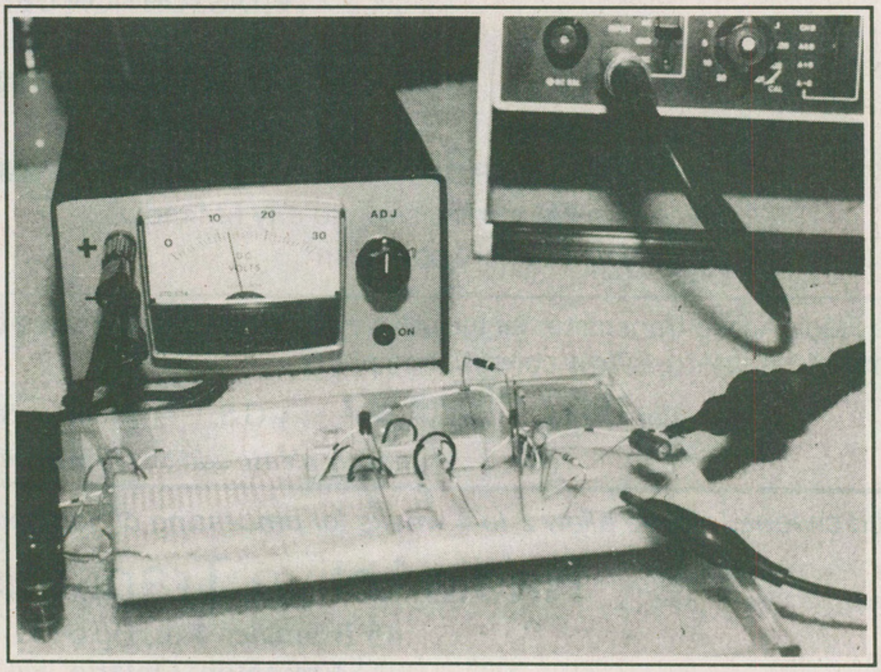


*Figure 2*

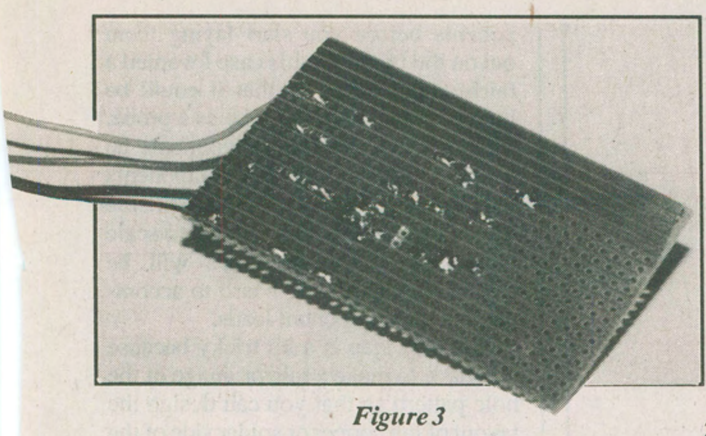

*Figure 3*

horizontal interference patterns every time the oscillator was turned on.)

If you are beginning to think that the fabulous, famous transistor we have heard so much about for years is not as wonderful as was originally thought, you are right. Transistor design is difficult and tedious - but it was an improvement over tubes and such. Overall, it is much simpler to design an amplifier using op amps or to build an oscillator using a chip produced for that purpose - and we'll get to that eventually in this series.

But, hey... isn't this fun?

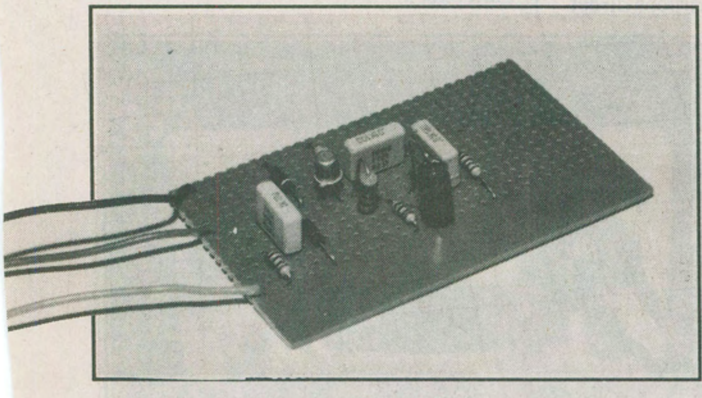

*Figure 5*

Well, let's build it anyway.

The components used in the circuit are shown on the schematic and are available at any electronics supplier. The transistor I used was a 2N2222A. This is a common small signal transistor in a TO-18, metal can package. You could use a variety of NPN transistors in this configuration, but make sure you know the pin configuration when you set it up as they differ somewhat.

(Here is the most useful hint I'll give you in this series: Get yourself an ECG Semiconductors Master Replacement Guide from an electronic supplier who handles Sylvania components. It is a very useful reference for a wide range of parts, specifications, pin configurations, etc. It is also a good cross reference from one manufacturer to another.)

But, back to the transistor we are using. If you cross reference the 2N2222A with the ECG book it goes to an ECG123A. The specs show the power dissipation as .500 watts or one-half watt. We'll talk about specs in another articles but, for now, as long as you don't load the output of the oscillator down too much in use you should not have any problems. If you do damage the transistor, it's cheap to replace anyway. The other spec we were concerned about in designing the amp was beta, or $h_{FE}$. This is given as 200 typical. We minimized the amplifier's dependency on this by bypassing only one part of the emitter resistance, $R_E$, with capacitor, $C_B$. $R_e$, which is unbypassed, swamps out most of the amplifier's dependency on the beta of the transistor. Other specs were used to make sure that maximum voltage and currents were not exceeded.

You could replace the 2N2222A with a number of other transistors. An ECG123AP, 2N3904 or 2N4401 should work with no problems. These are similar to the 2N2222A but come in a plastic TO-92 package and have lower power dissipation specs.

Figure 2 shows a photograph of the breadboarded circuit, powered by a twelve volt power supply and the components of the oscillator. A breadboard such as this is practically indispensable for testing out projects, is easy to use, and allows you to make changes easily. For example, on the breadboard I replaced the 10 ohm unbypassed emitter resistor with a potentiometer. Adjusting the pot allowed me to vary the gain to see at what point the circuit would begin to oscillate. I then measured the value of the pot and replaced it with a resistor. (I know, it's

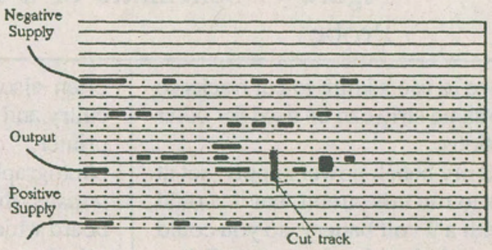


Figure 4. Solder side layout of Oscillator

cheating...you're suppose to be able to predict all that on paper ... don't tell my students.)

After I got the circuit operating on the breadboard I found a scrap of Veroboard and mounted the components on it. (Figures 3 and 4 show the copper side, Figures 5 and 6 show the component side.) Veroboard has parallel copper tracks and a matrix of predrilled holes to mount the components. The layout shown may help you to mount the components but actually this circuit is relatively simple and you would have no trouble setting up the layout yourself. Be careful when soldering to avoid solder splashes that will short out parts of the circuit. You

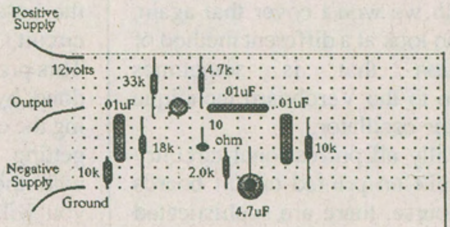


Figure 6. Component layout for Oscillator

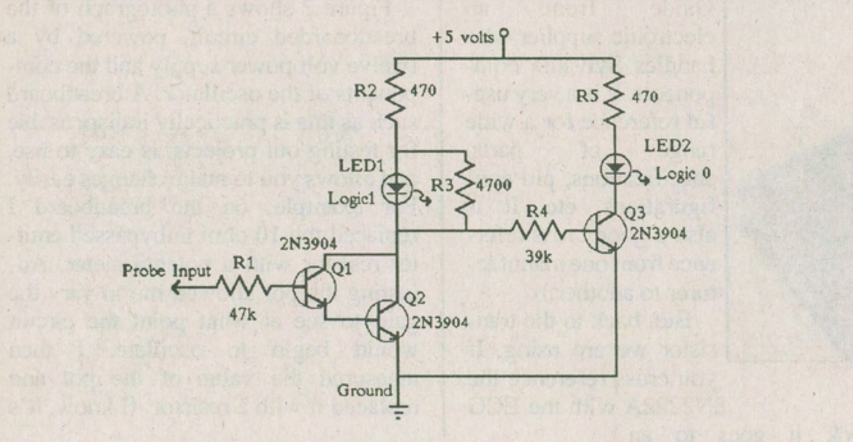


Figure 7. Schematic of a Logic Probe

may have to use a knife to cut tracks so that the same strip can be used for other connections.

I used my bench power supply, set at 12 volts to run this circuit, but it should work with a 9 volt battery, so you could build the whole thing into a small box or tube and use it as a signal probe. The circuit only has three connections: plus and minus power supply and output.

The other circuit we saw last month was a logic probe. Shown in Figure 7, the logic probe uses three small signal transistors as switches to sense the logic level (0 or 1, about zero volts or about 5 volts) and to switch on and off two light emitting diodes which indicate which logic level is present at the input. This can be a useful piece of test equipment when checking digital circuits, especially when an oscilloscope is not available. Again, this circuit is small and simple to build, and since it derives its power from the circuit under test, no power supply needs to be built. It can also be built into a small box or tube making it handy as a test probe. Since we went through the circuit operation last month we won't cover that again, but we can look at a different method of construction that is sometimes preferable to the Veroboard technique used for the oscillator.

Practically all professional circuitry is now built on printed circuit boards and, of course, there are sophisticated techniques used to go from the original design to a finished printed circuit board. Computer programs which help in the schematic design process can often also be used to lay out the circuitry and then generate artwork with a printer or plotter. From there photographic techniques are used to transfer the design onto a copper clad board which, when etched using chemicals, becomes the final circuit board.

There are kits available which can set you up to use this process with some variations. You can use direct artwork transfers which bypass the photography allowing you to layout the traces directly on the copper clad board and then etch it. The cheap and dirty approach, (which works satisfactorily for simple circuits), is to directly draw the traces onto the copper using a non-water soluble felt pen. That's what I used for the prototype of the logic probe.

Figure 8 is a photograph of the breadboarded circuit, a layout pattern and the un-etched printed circuit board itself. This procedure is best done by breadboarding the circuit first and getting it working with the components you will use on your pc board. That way you will know the physical sizes and shapes of all components before you start laying them out on the board. In this case I wanted a fairly small board so that it could be built into some kind of tube as a probe. This required that the components be close together. Once the components are laid out properly a component side layout pattern should be drawn to scale indicating where the holes will be drilled through the pc board to accommodate the component leads.

The next step is a bit tricky because you have to make a mirror image of the hole pattern so that you can design the layout of the copper or solder side of the pc board. See Figure 9. (Sounds simple but this usually causes some problems.) Once you have the pattern of holes for the solder side the designing of the track layout to connect the components leads as in the original circuit is done. Sometimes you are forced to go back and change the component layout to allow interconnections. You may even need to add a jumper wire to the design of the top of the board to facilitate the connections. (In this case it was simple enough that no jumpers were necessary.) When you finally have the design laid out you will draw the circuit onto the copper clad board.

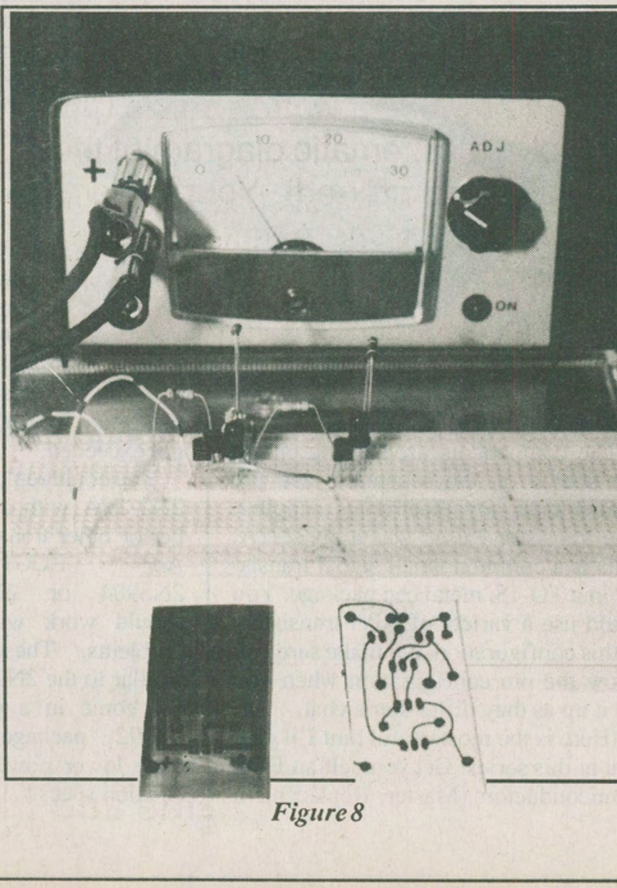


*Figure 8*

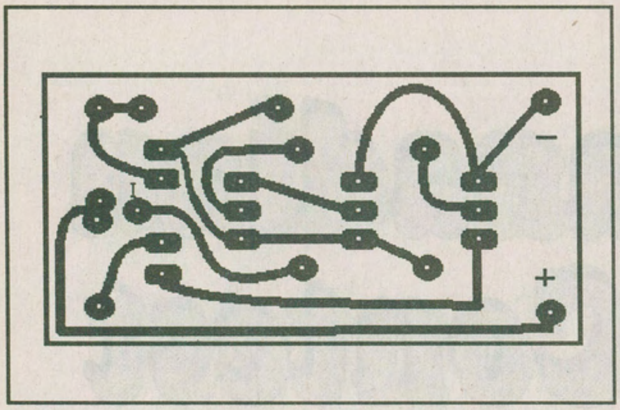


*Figure 9. PC Layout for Logic Probe*

Copper clad board can be obtained from an electronic supplier in a variety of qualities. The thickness of the copper can vary with heavier boards being less likely to lift the tracks and pads when you solder to them. The board itself comes in epoxy or phenolic, primarily. Epoxy is more expensive and stronger while phenolic is less expensive but brittle and may break if dropped. I prefer epoxy but for most applications phenolic is acceptable.

Before you start drawing pretty traces on the copper side of the board the copper must be cleaned of all oxidization. If you don't the etchant you use later will not work evenly. One way to do this is by scouring the surface with steel wool. A scouring pad and fine sand works well but most people will not have easy access to it. In a pinch fine sandpaper would be acceptable but may be difficult to maintain an even amount of abrasion causing too much copper to be removed. The object is to make the copper look shiny without taking off too much of it.

The next step is to place the printed circuit layout over the copper side of the board and mark where the holes will be drilled. This can be done by lightly centre punching each pad drawn on the layout, which leaves an indentation to indicate the layout and provide a dimple for drilling later.

You are now ready to draw the circuit traces on the board. The main concerns here are to avoid getting fingerprints or other contaminants on the board (which may interfere with the etchant) and to draw the circuit traces as cleanly as possible leaving enough space between them so the etchant can separate them. The whole procedure is a bit tricky at first but you'll get onto it with no major problems. Remember: Use a fine tipped felt pen of the type that is non-water soluble. Pens sold specifically for this use are available at electronic suppliers.

Once the layout is drawn and dry, the board can be etched, and should be within the next few hours as the copper will begin to oxidize if you don't. There are a couple of etchant chemicals which are obtainable from your local electronic supplier. I use ferric chloride solution because, though it is slow, it will etch fairly well at room temperature and is relatively safe to use. (When you buy this obtain a Material Safety Data Sheet and read it to familiarize yourself with safe handling procedures, etc.) To use the etchant just pour it in a shallow tray, drop the pc board in with the copper side up and move it around every few minutes. Etching may take fifteen or twenty minutes depending on how clean your board is and how much copper must be etched off. The areas where you have drawn traces should be impervious to the etchant unless you leave it in too long. A bit of experimentation should perfect your technique.

When you are done etching wash the board under the tap (again, refer to the MSDS for safe handling instructions of the etchant). From here you can use the steel wool to scrub off the ink and your board is ready to be drilled, stuffed and soldered. Figure 10 shows the component layout for the board.

We'll talk more about printed circuit board techniques in subsequent articles as we get into more complicated layouts requiring photographic techniques. In the meantime this one is simple and yet useful and inexpensive if you have never tried it before.

That's all for this time. Watch for some more theory on semiconductors coming up. ❐

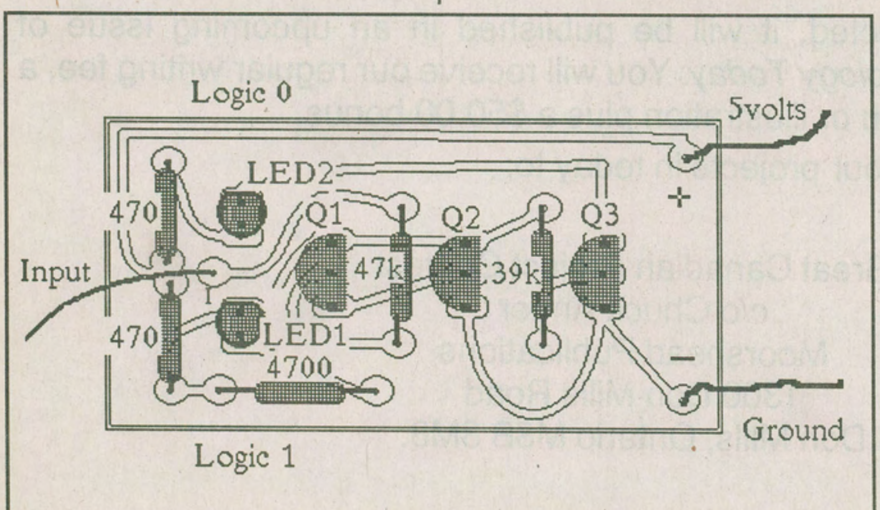


*Figure 10. Component Layout for Logic Probe*