

How To Build The Tremulous Bear

By Steve Daniels, President, Small Bear Electronics LLC

What Is The Tremulous Bear?

The Tremulous Bear is a boutique-quality tremolo. It does all of the good things you would expect of a modern, flexible trem: traditional Fender-style chop, subtle background quavering and weird modulations. The waveform generator is an interesting design; you get choice of triangle or square wave and the ability to set the “on” and “off” intervals.

This manual contains complete instructions for building the pedal, and it is written to guide people who have never built a pedal before. (Experienced hobbyists note: You can skip sections as you need to.) I don't presume that you know any electronics, but you do need some skill with hand tools and I don't cover in here how to solder; more about these issues in the next section. That said, much of the information and many of the techniques shown here are applicable to building *lots* of other pedals, and even other electronic devices.

I know that you'll enjoy building--and playing through!--the Tremulous Bear. While I've done my best to make these instructions complete, I'm available by E-mail if you have questions, problems or suggestions.

Yours In Good Music,

A handwritten signature in black ink that reads "Steve Daniels". The signature is written in a cursive, flowing style.

Small Bear Electronics LLC
123 Seventh Avenue # 156
Brooklyn, NY 11215
[Http://www.smallbearelec.com](http://www.smallbearelec.com)
Smallbearelec@ix.netcom.com

Can I Really Build It Myself?

Yes, IF:

- ✦ You can follow directions.
- ✦ You are comfortable with using basic hand tools.
- ✦ You can solder well.

IMO, the hardest part of the assembly is not the electronics; it's preparing and decorating the case. I will show you in this manual, step-by-step, the tooling and decal application procedures that I used. If you have never soldered before, you may need to check out some how-to information on-line about that, and maybe practice a little before tackling this kit. Which brings me to:

What Tools And Materials Will I Need?

You need a drill, or access to one, and either a set of twist drills or a step drill like a Unibit. If you want to install the battery drawer, you should either have a Dremel, or similar, rotary tool, or have access to one that you can borrow. You will also need:

A 25- to 35-watt soldering iron, rosin-core solder and cleaning sponge.

NB: Yes, a Radio Shack iron will do, if the tip is relatively new and well-tinned. The big problem with very inexpensive irons is that the tips aren't properly clad and so corrode quickly.

Small screwdriver

Small chain-nose plier and side-cutter

Small locking-grip ("Vise-grip") plier

X-acto or similar knife

Steel Rule

Self-locking tweezers or other "third hand"

Colored pencil or "Hi-liter" felt-tip marker

Some small round and flat files

A pointed steel "pick" or scratch awl












De-soldering braid (like Radio Shack p/n 64-2090B)

A ready-to-apply decal comes with the kit. Sealing this requires a clear, spray lacquer like Krylon. You will also need some epoxy cement for attaching the battery drawer to the enclosure.

◆ Identify The Components

5% Tolerance Carbon Film Resistors

The values are in Ohms. The first two colored bands define the base number. The third band is the number of zeroes to add, and the gold fourth band indicates 5% tolerance.

		How Many?
1,000,000 ohm or 1 Megohm (Brown, Black, Green, Gold) 1 0 + 5 zeros		1
2,700 ohm (or 2.7 K ohm) (Red, Violet, Red, Gold)		1
3,300 ohm (or 3.3 K ohm) (Orange, Orange, Red, Gold)		1
5,600 ohm (or 5.6 K ohm) (Green, Blue, Red, Gold)		1
22,000 ohm (or 22 K ohm) (Red, Red, Orange, Gold)		3
33,000 ohm (or 33 K ohm) (Orange, Orange, Orange, Gold)		1
47,000 ohm (or 47 K ohm) (Yellow, Violet, Orange, Gold)		1
100,000 (or 100 K ohm) (Brown, Black, Yellow, Gold)		6
150,000 ohm (or 150 K ohm) (Brown, Green, Yellow, Gold)		1
470,000 ohm (or 470 K ohm) (Yellow, Violet, Yellow, Gold)		1
100,000,000 or 100 Megohm (Brown, Black, Violet, Gold)		1

Potentiometers

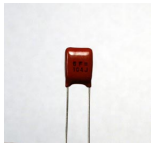
This style has pins that solder directly to the PC board.



Quantity	Marked	
500K Reverse Audio	2	C500K
1 Meg Reverse Audio	1	C1M
10K Linear	1	B10K
50K Audio	1	A50K

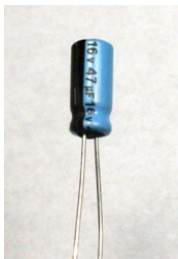
Capacitors

Metallized Polyester Film (Has no polarity)



Quantity	Marked	
.01 mf. (microfarad or μ .)	1	103
.1 mf.	1	104
.22 mf.	1	224
.68 mf.	1	684

Radial Electrolytic (These are polarized; the black band marks the negative side.)



	Quantity
10 mf.	2
100 mf.	2

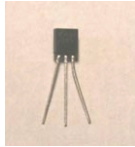
Tantalum Electrolytic (This is polarized; the short lead is the negative side.)



	Quantity
2.2 mf.	1

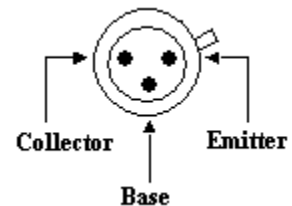
Transistors

Two of the transistors come in a plastic case called TO-92 style. The flat side of the case will be oriented according to the photos of the board and its silkscreened legend. The designations of the leads are shown for reference.



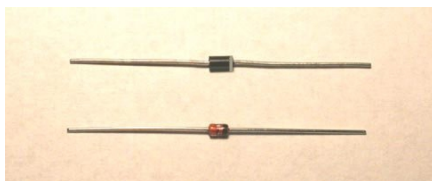
	Quantity	Leads Left-To-Right
2N7000	1	Gate, Source, Drain
MPSA14	1	Emitter, Base, Collector

The 2N2222A transistor is in a metal TO-18 case, and the legend on the PC board indicates where the tab on the case must be positioned.



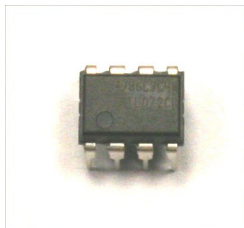
Diodes

The band on the case indicates how they should be oriented on the PC board.



	Quantity
1N5818	1
1N914	2

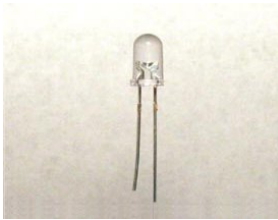
ICs



Both of the ICs in the Tremulous Bear come in this 8-pin Dual In-Line (DIP) package. The engraved dot marks pin 1. Some makers also put a notch on the left side to indicate this.

	Quantity
TL072	1
TS555	1

LED



The longer lead marks the positive side.

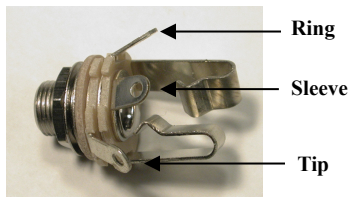
Photocoupler



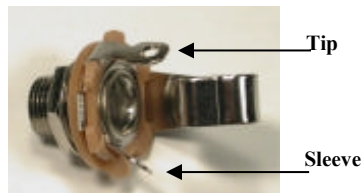
The dot on the left side marks the negative side. The picture of the PC board shows how this component is oriented.

Jacks

Stereo, Open-Circuit, 1/4"
This is the input jack.



Mono, Open-Circuit, 1/4"
This is the output jack.



Toggle Switches



Stomp Switch



Other Items

IC Sockets



Plastic LED Holder



9-Volt Battery Snap

Enclosure

Battery Drawer

Rubber Feet

PC Board

Knobs

Decal

Hookup Wire

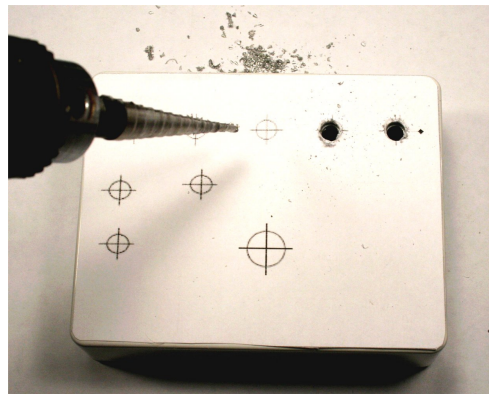
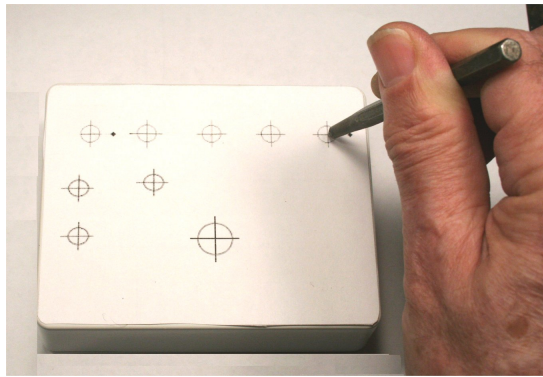
If you have all the parts and materials, the next step is tooling and decorating the case.

Tooling The Case

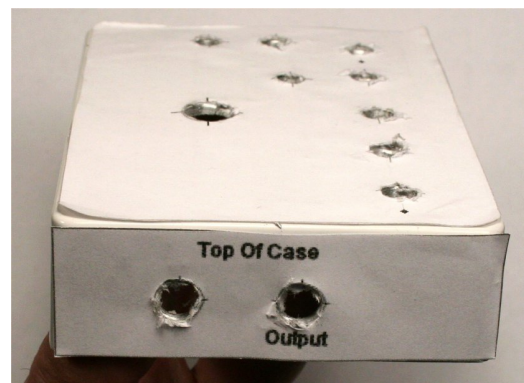
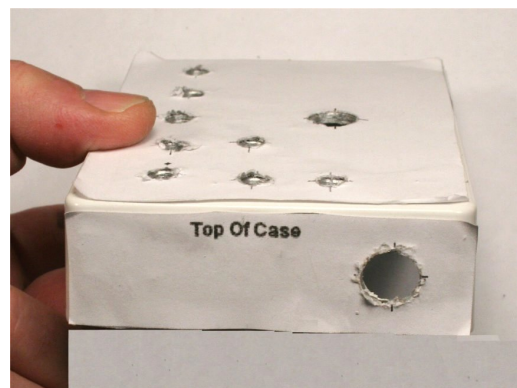
The enclosure has been powder-coated. Your job is to drill holes for the pots, stomp, switch, jacks and in-use LED. Then you can either apply the decal or otherwise add labels to the controls.

First, download the drilling template. If it does not print at actual size, open it with a program like PaintShop that will let you re-size it to 600 x 800 pixels at 72 pixels per inch. With a sharp scissor, cut out the piece for the top of the case. **Cover the top with masking tape first;** that will help to keep the finish from cracking. Then, using double-sided transparent tape, attach the template to the case.

Using a scribe, center-punch or pick, mark the centers of the holes for the pots, LED, stomp switch and toggle switches and drill them. I used a Unibit for this, but regular twist drills are also fine. The hole sizes are: pots, 9/32"; toggle switches, 1/4"; stomp switch, 1/2"; LED holder,



In a similar way, fasten the templates and drill the holes for the DC power jack and the input and output jacks.

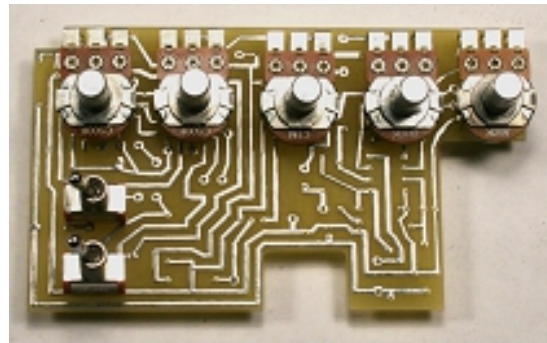
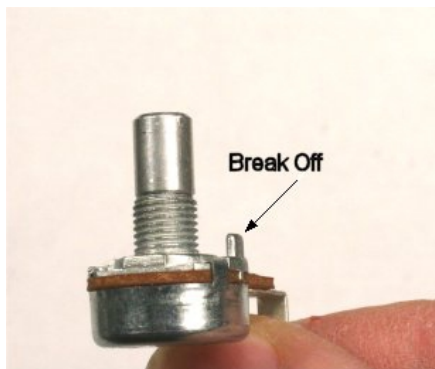


With all the drilling done, strip the templates and tape.

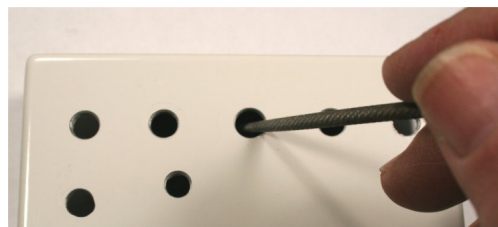
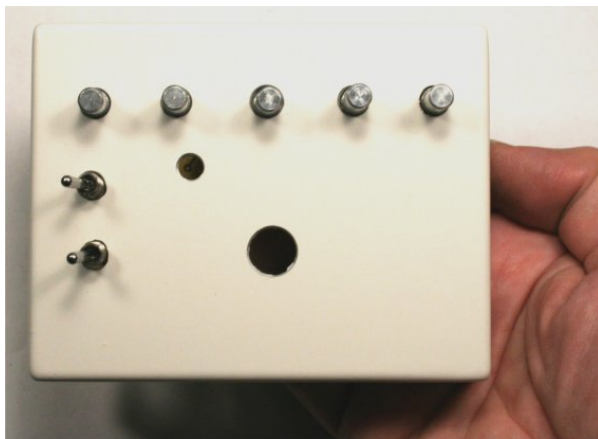
Before continuing, this is a good time to grind off the powder coating where the input and output jacks sit so that they get a solid ground to the enclosure. This can be done with sandpaper or a file, though it is a bit of work. A grinding stone in a drill or Dremel handpiece is much more efficient.

Now you are ready to “mock up” the assembly of the pedal to make sure that the registration of the holes is correct.

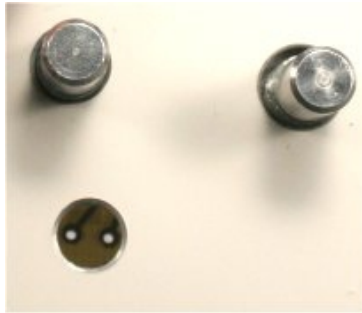
Remove all of the nuts and washers from the pots and the switches. Using a diagonal cutter, break off the anti-rotation tab from each pot; this build will not use them. Then temporarily install the controls and switches in their appropriate holes on the board.



With your fingers on the underside as shown in the next pic, slip the board assembly up through the drilled holes. If the registration is off, one or more parts will clearly feel like they are binding--going in off-center. Note any problem areas carefully. If you have done an otherwise careful job, a little work with a round file will create enough play to let everything to go in straight.



Looking straight into the hole for the LED, you should be able to see the two holes for its leads.



Once the board goes into the case smoothly, slip the washers and nuts onto the pots and switch and finger-tighten. When tightening the nuts, be sure that none of the components is being pulled out-of-true. If you notice any of that, disassemble and go back to work with the file

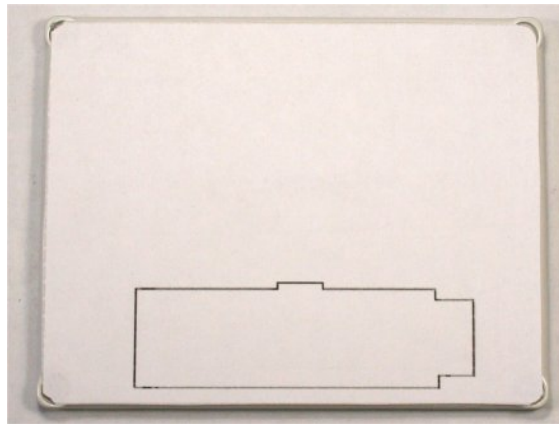
To check their hole sizes, temporarily install the DC power jack and the input and output jacks. If everything now looks like this:



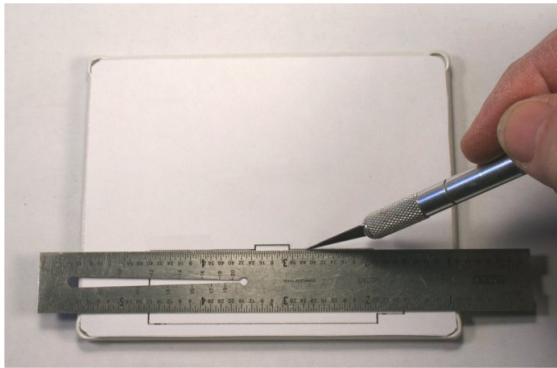
you are ready to create the opening in the lid for the battery drawer.

If you don't want to install the battery drawer, you can skip this section and simply add a spring-type battery clip later. Installing the drawer is a good deal of work, but you get to change the battery later on by popping the door open.

Begin by downloading the template from www.smallbearelec.com/Projects/TremBear/Kit/BattDrawer.jpg, cutting it out and fastening it to the top using double-sided tape.



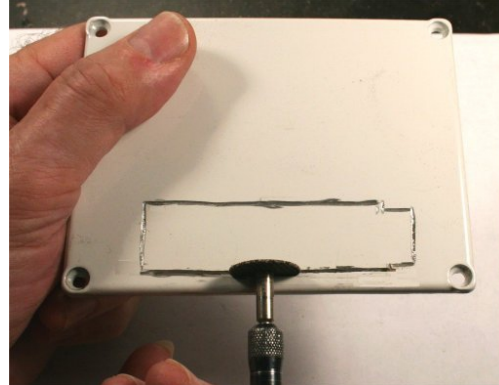
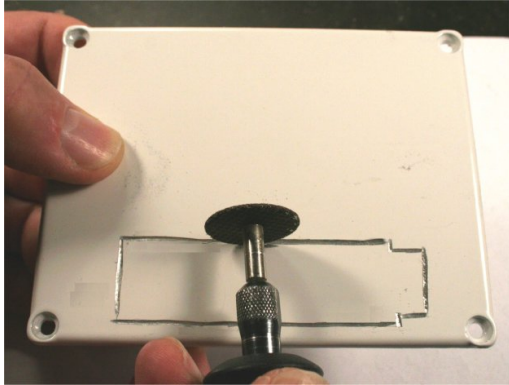
Using an X-acto or similar knife with a fresh blade, and a steel rule, score all along the outline of the cutout. When done, you can remove the template.



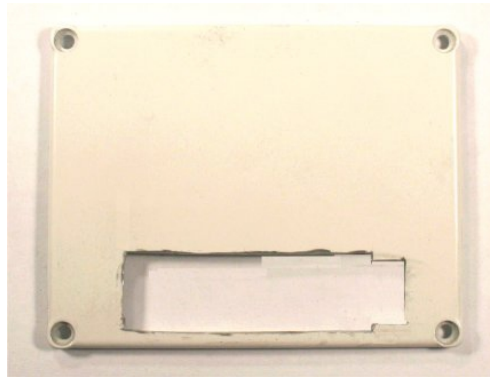
It is possible, though very laborious, to cut this opening by drilling many small holes and then filing to size. If you don't have a Dremel tool, use this project as a reason to buy or borrow one. Cutting the opening requires a cutoff wheel, which comes with most Dremel accessory sets.

Note: **Abrasive cutoff wheels are both very useful, and inherently dangerous! In normal use, they throw off particles of metal and abrasive. When they break, and they do, the pieces fly like bullets. You MUST wear eye protection and a dust mask when using them!**

Start the cutout by marking the outline. Then go back and do the definitive cuts.

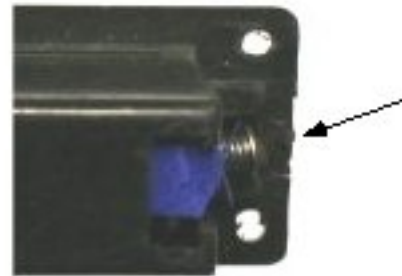
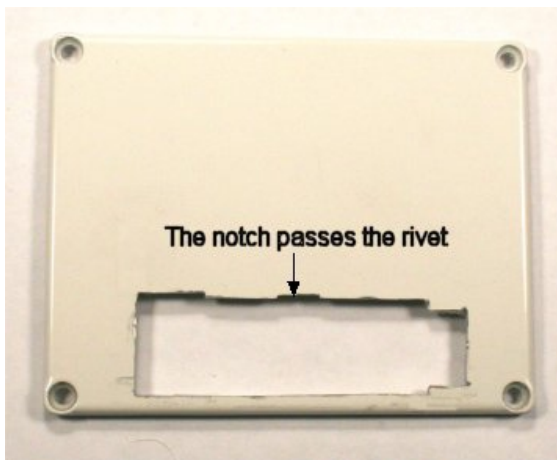
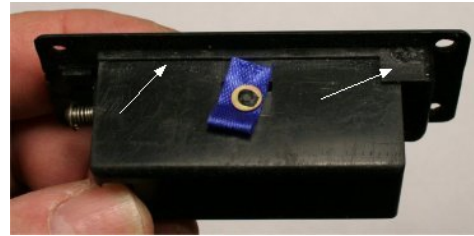
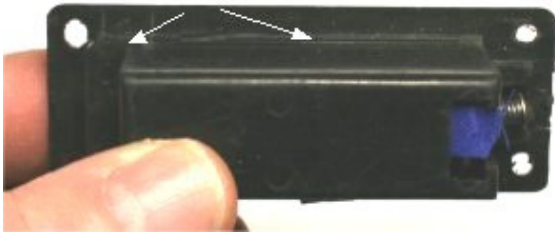


Take your time. It's normal for the piece to get hot from friction, and you'll have to change the cutoff wheel at least once. Take care not to let the tool slip. If you are patient, you'll wind up with a rough cutout.



Don't be scared when you find that the battery drawer doesn't quite go down all the way in the opening. Its housing is not at all perfectly rectangular, so you'll need to do some work with a flat file to accommodate the ridges and the rivet shown in the pics. With a little file-and-try, the battery drawer will fall into place easily.

In order for the door of the battery drawer to spring open, the latch that secures it must be able to slide slightly rightward. To permit this, grind down the small nib so that it can't rub against the surface of the enclosure.



With all of this done, the battery drawer should fit comfortably. Hold it in place and be sure that the door snaps open easily. You'll secure it later after the case is decorated.



The next step is decorating the case.

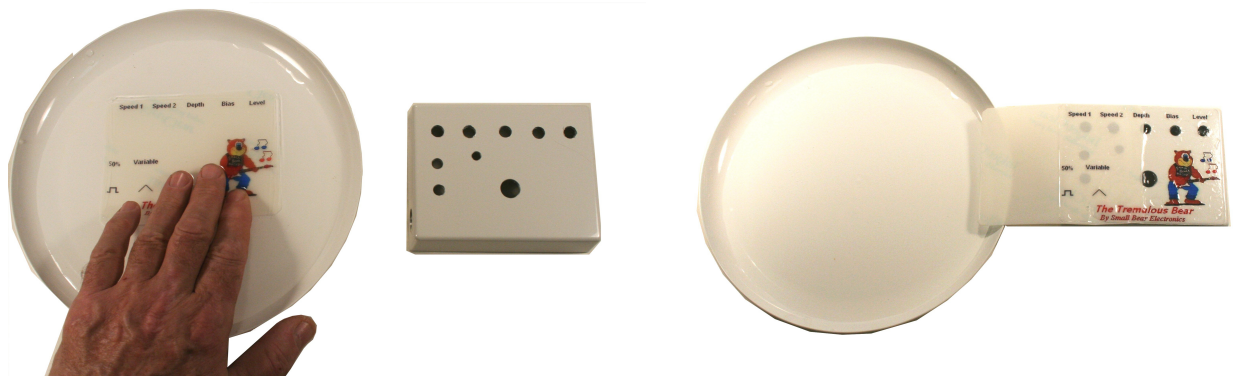
Decorating

If you want to paint, you'll have to seek a different reference. This manual will walk you through applying the decal that comes with the kit. You can also create your own artwork if you don't like the Small Bear decal, and we'll briefly discuss what is involved in doing that.

Remove all of the panel-mounted components: pots, jacks and stomp switch.

To prepare the surface, wipe the enclosure thoroughly with a soft cloth wetted with denatured alcohol. Read and follow all the precautions on the can. Let the surface dry thoroughly.

The decal that comes with your kit was printed on an ordinary inkjet printer to "water-slide-release" decal stock, and then sprayed with clear lacquer to protect the image. Cut the decal to size with a sharp scissor. Set up a small dish with some warm water. Place the decal in the water, and hold it under gently with your fingers. Within a minute, you will start to feel the decal film slide away from its backing. At that point, remove the decal from the water, position it on the surface of the box, and carefully slide the backing paper out from under the decal film.



While the film is wet, you have some control of its exact position. Position the decal where you want it, and then blot the surface firmly and thoroughly with a paper towel. You want to squeeze out All of the water and get rid of air bubbles. When done, let the decal set for several hours. To protect the design, finish with a clear lacquer like Krylon or an automotive clear coat. Spray a very light first coat, and give that a couple of minutes to get tacky before putting down additional coats. These steps are a little tricky, but once you know how the material feels and works, you can get some remarkably good results repeatably.

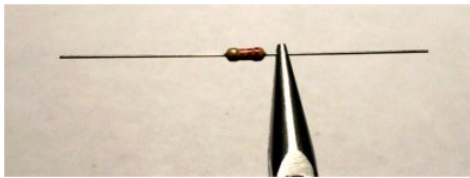


The enclosure is finished, and you are ready to assemble the PC board.

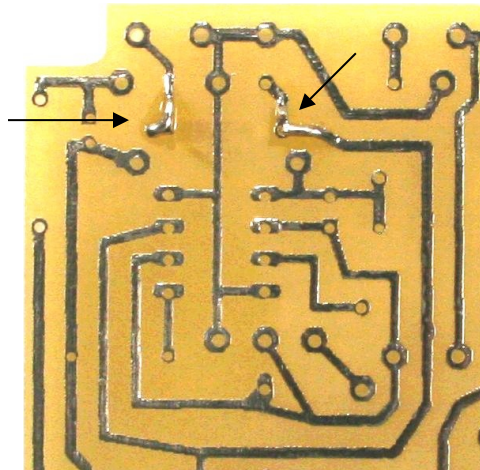
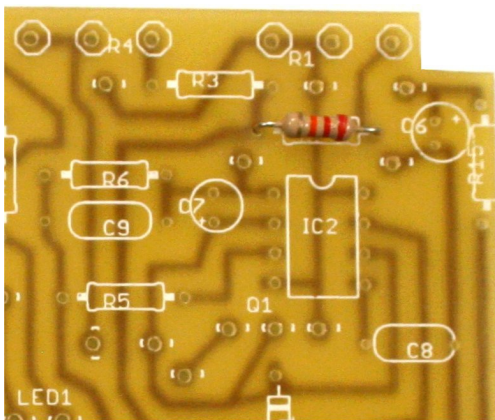
Stuffing The Board

That's what we call the process of mounting components on the printed circuit board (PC board) and soldering them in place. While the order in which the parts are mounted won't affect the audible result, it's convenient to mount small parts first and finish with the pots and toggle switches. Heat up and tin your soldering iron, wet your cleaning sponge and let's begin, starting with the resistors. Find resistor **R2 (22K = Red, Red, Orange, Gold)**

Using a chain-nose plier, grab one lead of the resistor about 1/16" from the body. Bend the resistor lead sharply downward at right angles to the body of the component. Then do the same with the other lead:



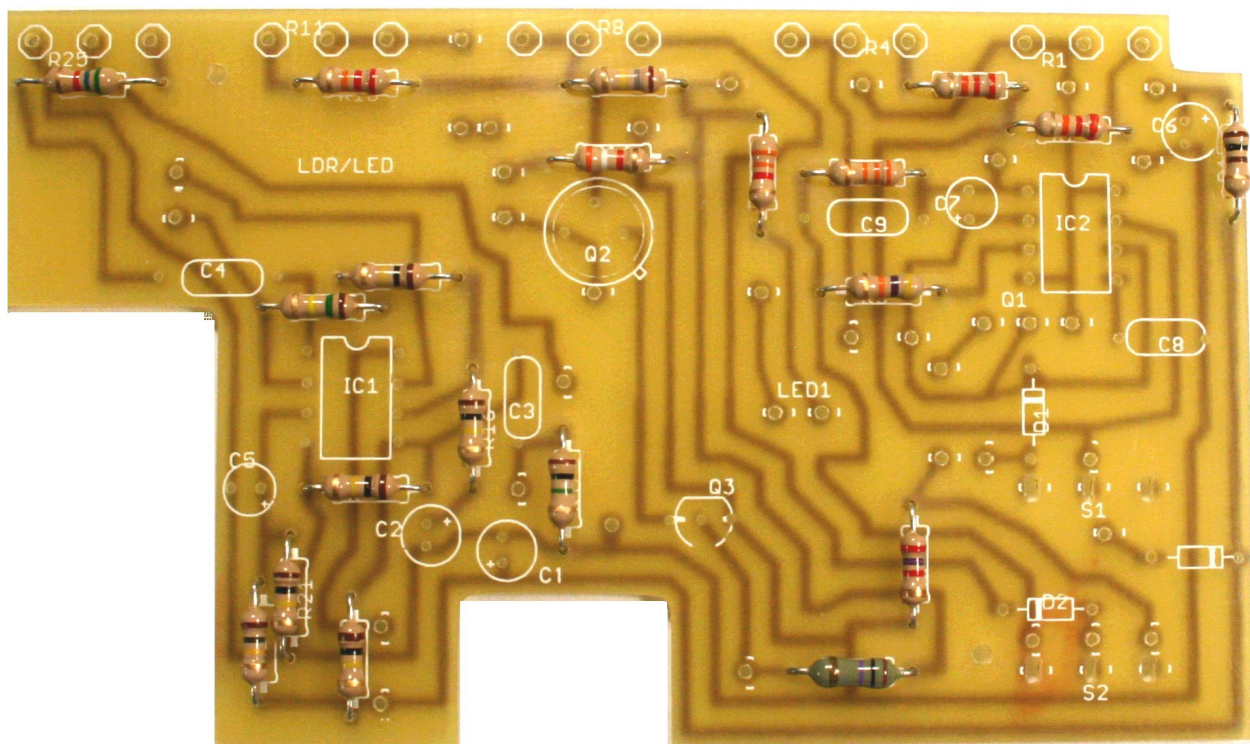
Look for the silkscreened legend "R2" on the PC Board, and insert the resistor into the appropriate holes. (Resistors have no polarity and can go in either way.) While holding the resistor in place, cut off all but about 3/32" of the leads and bend flush to the board. Solder in place. Bending the ends of the resistor leads flush to the board is very important in this build, because the solder side of the board mounts directly over the cans of the potentiometers; keeping the lead profile low will avoid short circuits.



In similar fashion, form the leads for the remaining resistors and solder them in place. They are:

- R3 (22K = Red, Red, Orange, Gold)**
- R5 (47K = Yellow, Violet, Orange, Gold)**
- R6 (33K = Orange, Orange, Orange, Gold)**
- R7 (2.7K = Red, Violet, Red, Gold)**
- R9 (180K = Brown, Gray, Yellow, Gold)**
- R10 (22K = Red, Red, Orange, Gold)**
- R12 (100 Meg = Brown, Black, Violet, Gold)**
- R13 (3.3K = Orange, Orange, Red, Gold)**
- R14 (May vary, will be bagged separately)**
- R15 (100 Ohms = Brown, Black, Brown, Gold)**
- R16 (100K = Brown, Black, Yellow, Gold)**
- R17 (1 Meg = Brown, Black, Green, Gold)**
- R18, R19, R20, R21, R22 (100K = Brown, Black, Yellow, Gold)**
- R23 (150K = Brown, Green, Yellow, Gold)**
- R24 (5.6K = Green, Blue, Red, Gold)**

Here's the board at this point:



Capacitors come next. Start with C6, which is a **100 mf. electrolytic**. The side with the black band is negative. Note its position in the pic and be sure to insert it correctly. Press its body close to the board and trim the leads down to about 3/32". Bend the leads flush and solder. Follow with C1, also 100 mf. electrolytic. Again, observe the polarity marking; the side with the black band faces the outline for capacitor C3. Next in are the 10 mf. electrolytics C2 and C5.

Capacitor C7 is a tantalum electrolytic and is polarized. The negative side (short lead) goes closest to the notch in the outline for IC1.

The rest of the capacitors are polyester film types; they have no polarity. Take care to get the right values in the right positions. The designations and the values are:

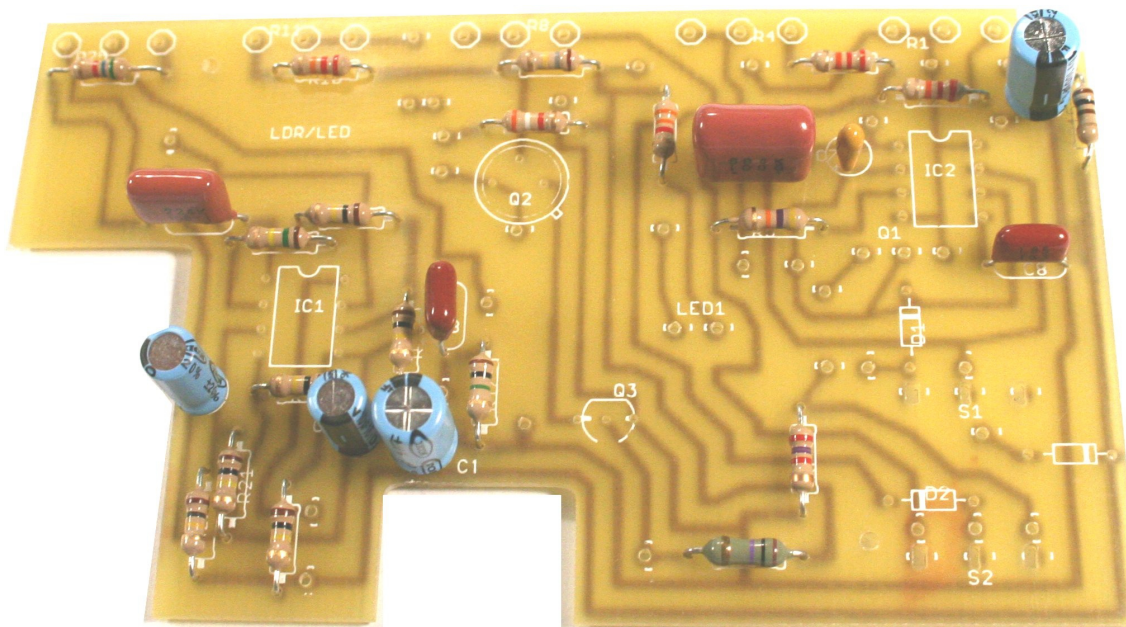
C3 - .1 mf.

C4 - .22 mf.

C8 - .01 mf.

C9 - .68 mf.

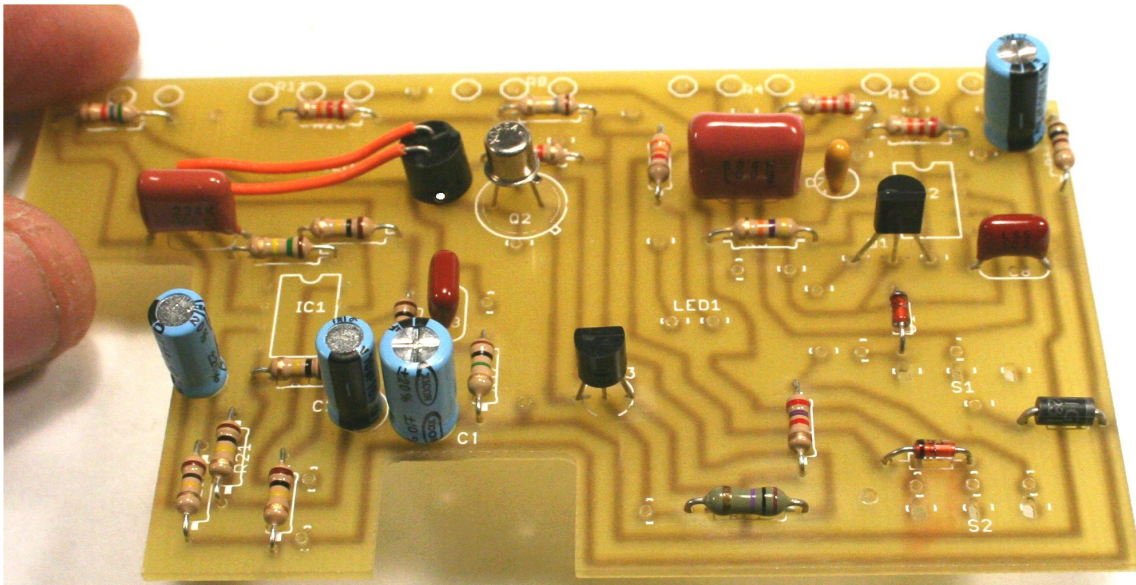
Here's a look at the board to this point:



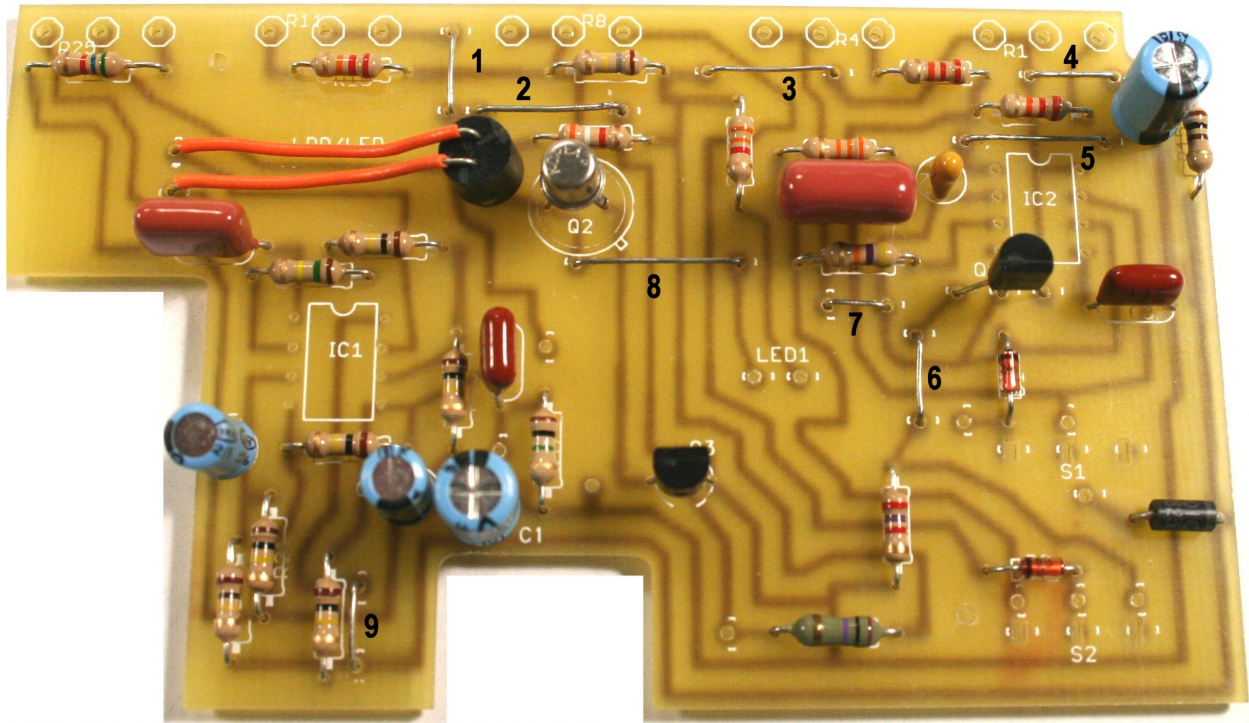
OK so far? Install the transistors, using the flat sides on the plastic ones and tab on the metal can device to orient them correctly. Be sure that the 2N7000 is Q1 and the MPSA14 is Q3. Modern silicon transistors can tolerate normal soldering heat without special protection. Still, it is good to know how to heat-sink a device when soldering. Insert transistor Q3, orienting its flat side to match the silkscreened outline on the board. On the bottom of the board, bend its leads flush to the surface. On top, clamp the locking tweezers onto the first lead to be soldered, between the board and the transistor body. You can also use an alligator clip for this job. Solder the first lead, then move the tweezers to the next lead, solder, etc. Do the same with Q1.

Follow with the diodes D1, D2 and D3, forming the leads as you did with the resistors. Observe polarity!: Match the bar on the case of the device to the silkscreened image on the board! The 1N914 signal diodes are glass packages, and the 1N5818 power diode is an epoxy case.

The Photocoupler includes in the same package the components labeled LED2 and LDR1 in the schematic, and you can see it at the upper right. Slip spaghetti tubing over its leads before soldering so that there is no danger of a short circuit. The board is tilted a little so that the dot that marks the negative side of LED2 is clearly visible. Take care to orient it correctly!

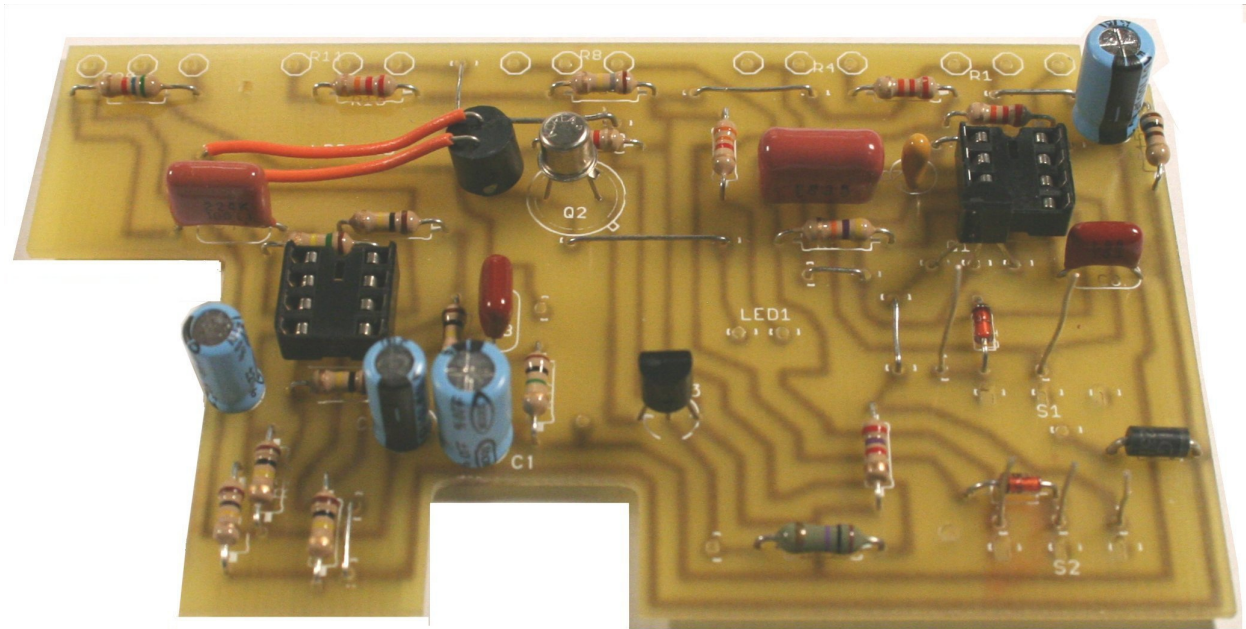


Rather than make the manufacture of a board more expensive by having lines on both sides, it is common practice to use short “jumpers” of bare wire on the component side to connect points in the circuit. Form these as you did the leads for some of the other components, and solder in place. There are nine, and they are shown in this pic.

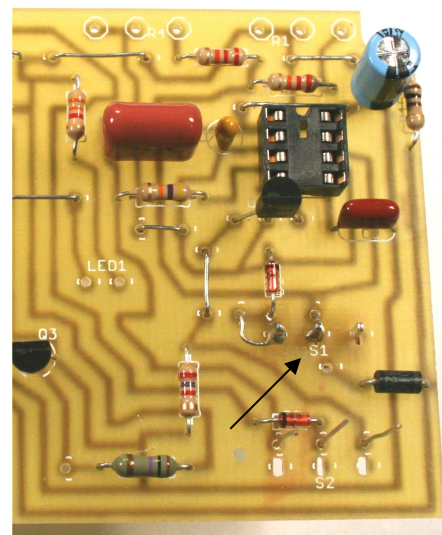
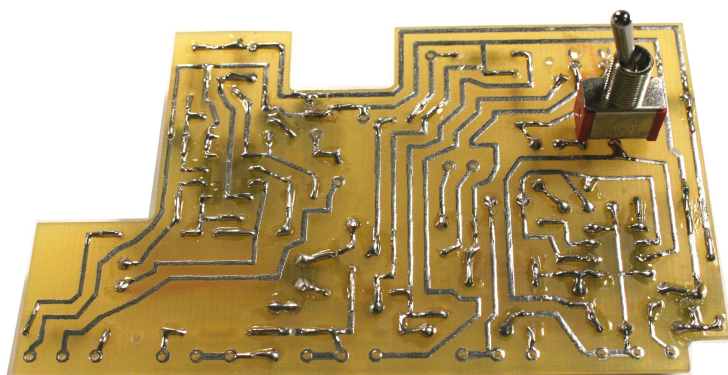


IC sockets are optional. I use them because they make it so much easier to replace a chip in the odd event that one does blow. Take care to line up the notch on the socket with the outline on the board. Use locking tweezers or other “third hand” to hold the socket while you solder.

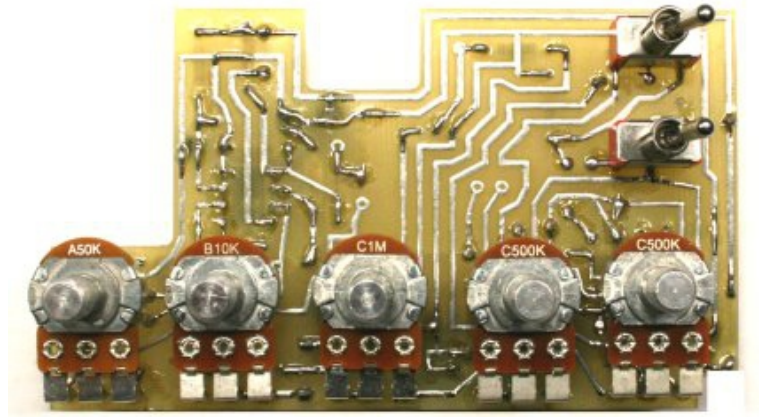
Prepare to mount the toggle switches by soldering short, bare wire connecting leads to the five pads shown in the lower right of this pic.



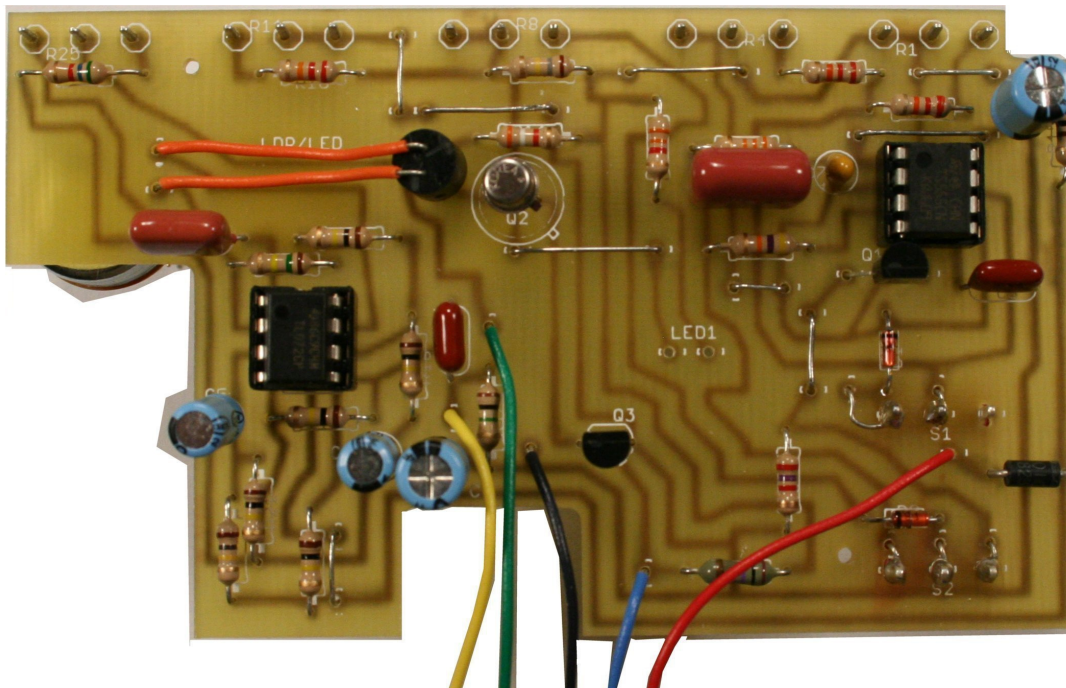
Remove the washers and nuts from the switches. Insert one switch into its holes and wrap the connecting wires around its pins. Before you apply solder, be sure that the body of the switch is vertical to the board and as close as possible to its surface. Solder in place, and then install the other switch in the same way.



Remove the washers and nuts from the potentiometers. Install the pots one at a time and solder in place. Pay attention to the lineup of values, and make sure that the contacts are flush to the board before applying solder.



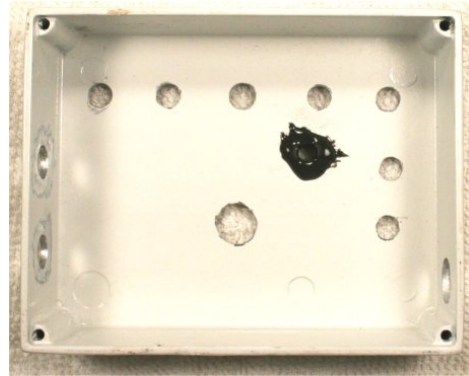
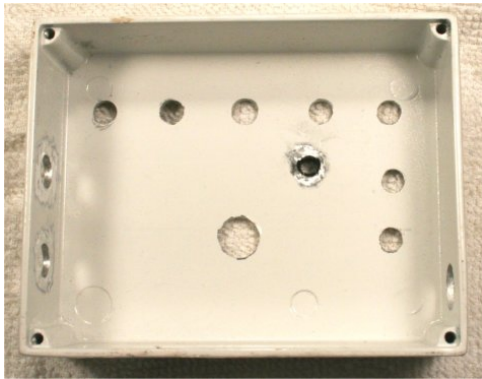
Before we do the initial and setup, plug in the chips, paying special attention to getting pin 1 in its right place. Then install five colored leads for: power positive (red), ground (black), PC board input, PC board output and LED control. In the pics that follow, input is yellow, output is green and LED control is blue. Use those colors, because I will refer to them when wiring the stomp switch later.



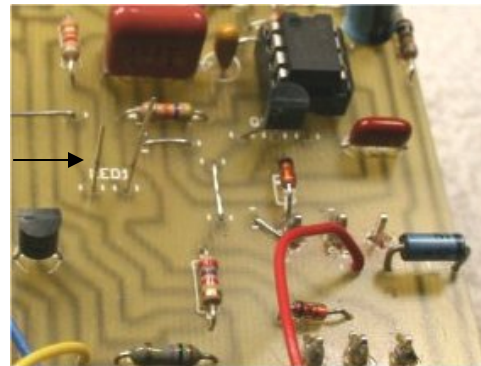
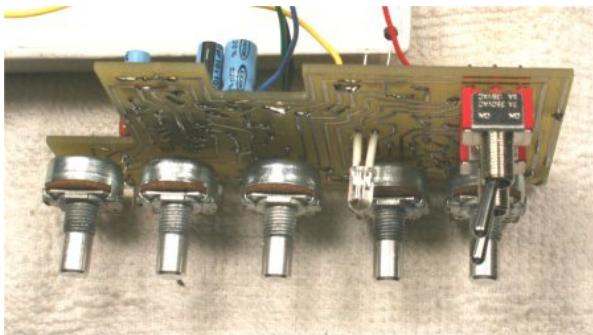
Putting It Together

Keep in mind that you are now at a delicate stage, because you have put a lot of work into finishing the case. Slips, falls and mistakes with tools can ruin the finish and cost you a lot of grief, so **stay focused and work patiently**. Set yourself up with a towel or soft cloth on your bench to protect the face of the enclosure while you work.

Grind away a little of the color around the opening for the LED. Position the LED holder in the opening. Mix a small amount of quick-setting epoxy cement and spread this around the ground area, making sure that it adheres to the flanges of the holder. Let the epoxy set for a couple of hours before proceeding.

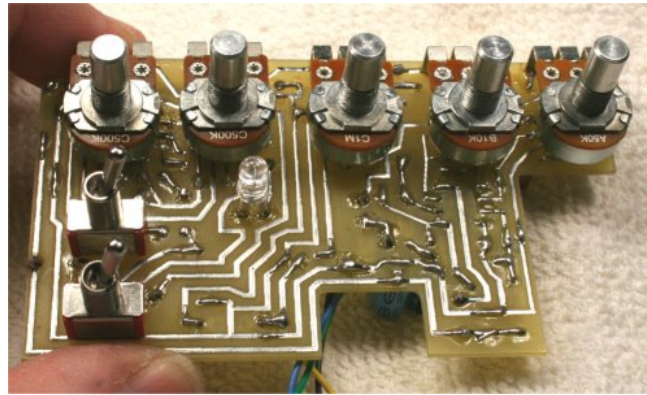
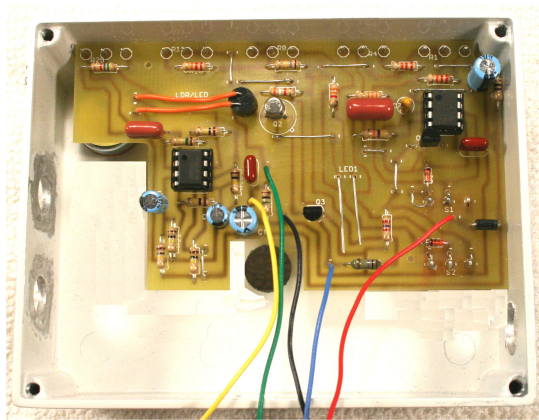


Slip a 1/4" long piece of spaghetti tubing on to each lead of the LED. Slip the LED into its holes, taking care that the short lead, which is the negative side, is on the left as in the right-hand pic below.



Now you get to find out how good your tooling really was. Slip the whole assembly into the case, lining up all of the components with their appropriate holes. You may have to jiggle things a little bit , and it is OK to push the LED downward to seat it in the holder.

When everything looks and feels “bottomed,” bend the leads of the LED down to the board to fix its vertical position as in the left-hand pic. Remove the works, and solder the LED in place on the solder side. The board is now finished, and we can do a preliminary test



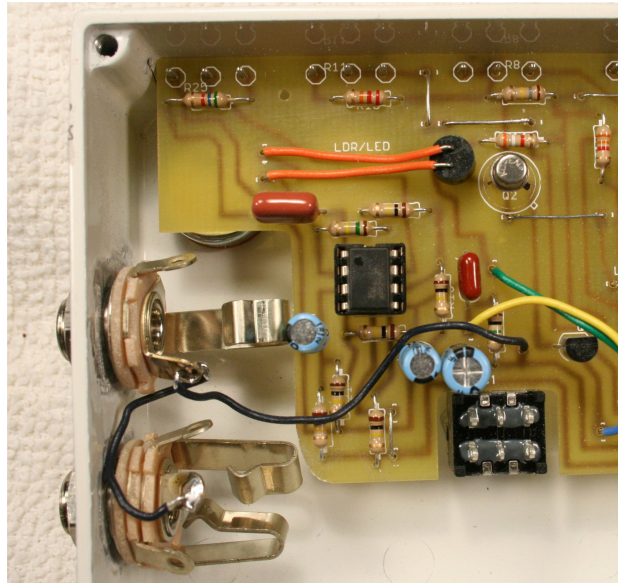
Attach the battery snap temporarily to the power leads from the board...+ 9 to red and – to ground. With the speed controls both fully counterclockwise and the bias and depth controls at about 30%, you should see the LED pulse slowly. It will have a sharp on-and-off with the waveform switch set to square and a soft ramp up and down in the triangle setting,

If you see this, the LFO is working and you can proceed with final assembly. If the LED does not pulse, you have some troubleshooting to do. Troubleshooting notes start on page 29.

Final Assembly

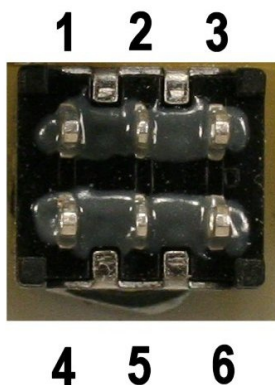
Install the input and output jacks, stomp switch and external power jack with their nuts finger-tight. Wire as follows:

- Run a lead between the Sleeve contacts of the input and output jacks, and add the black wire from the PC board ground.



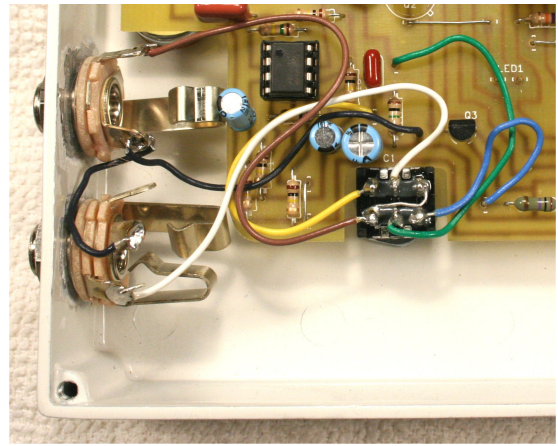
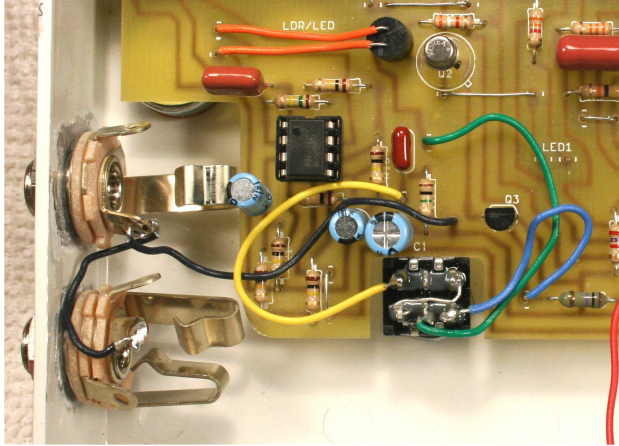
- Switch contacts:

This invariably confuses a lot of people, so let's do it "by the numbers."

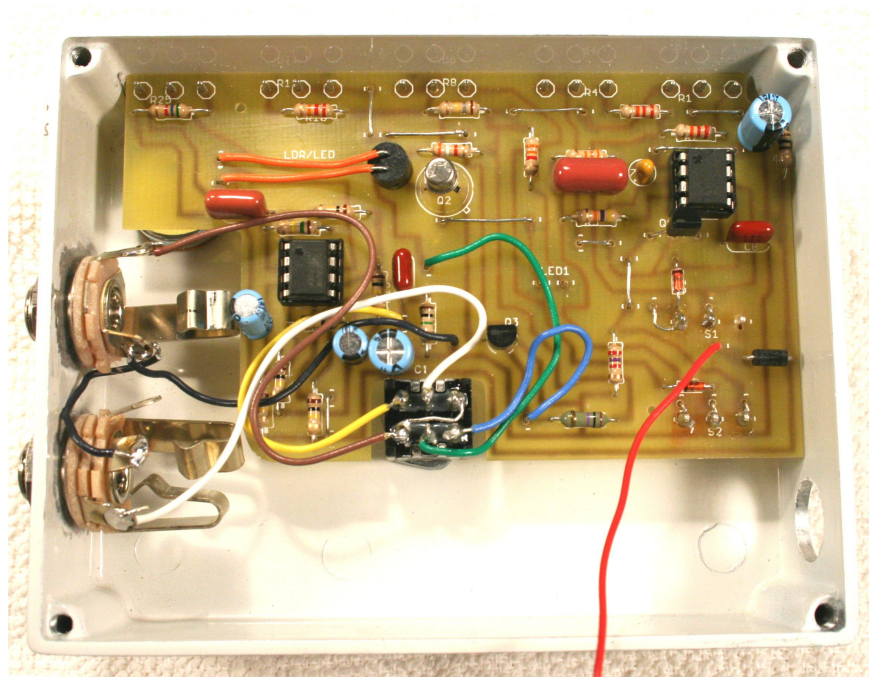


- #1 – PC board input (Yellow)
- #2 - Tip contact of input jack
- #3 - Connects to #4 and output jack tip
- #4
- #5 - To PC board output (Green)
- #6 - To LED control (Blue)

Start by connecting bare wire jumper from pin 3 to pin 4. Solder at pin 3, but not at pin 4 yet. Make the connection to pin 1 (yellow lead), then pin 5 and pin 6 (green and blue respectively.) Finish with a lead to pin 2 from the input jack tip and a lead from the output jack tip to pin 4. In the right hand pic, these are white and brown respectively.



Here's an overview at this point:



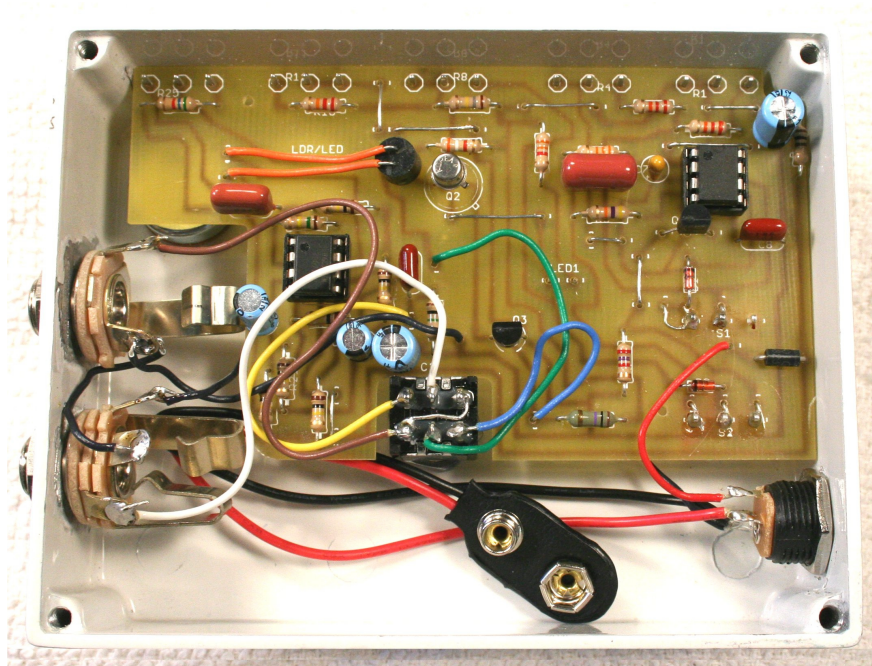
Since all of the wiring is done except for the power jack, you can now do a full-function test. Connect your guitar and amplifier, and temporarily connect the battery. The negative lead goes to the common ground at the sleeves of the jacks. An alligator clip lead will help in setting this up.

Bring up the Level control some and raise the Bias and Intensity. Check out the operation of the stomp switch. Got the effect? Controls work? Stomp switch works OK?

CONGRATULATIONS!

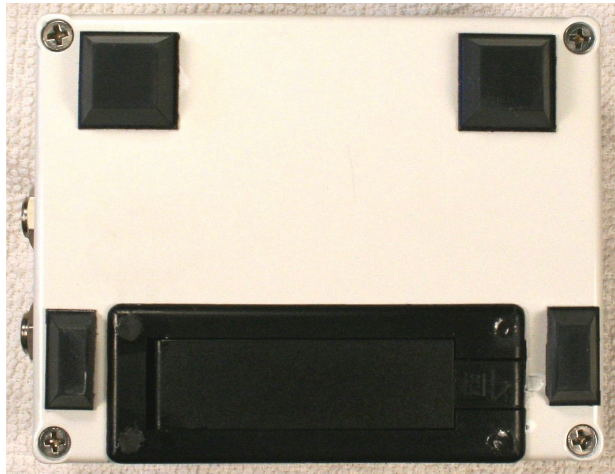
You have a working pedal. If something is not right, go to Troubleshooting. Otherwise you can finish up the wiring and assembly.

Run a lead from the Ring contact of the input jack to the ground terminal of the power jack, and join with the negative lead of the battery snap. Connect PC board positive to the + contact of the jack and battery + to the NC switched contact. The finished wiring is shown at right.



Tighten all the mounting hardware. ***Work slowly and carefully now!*** One slip of a tool and you'll be crying in your beer!

Fasten the battery drawer to the bottom plate with a large drop of epoxy cement in each of the mounting holes. If you have a Dremel tool and a small burr, you'll get better adhesion if you grind out the color at the bottom of each hole. Cut one rubber foot in half with a sharp knife. Attach the self-stick rubber feet to the bottom plate and screw this on. Return the pots to their full counterclockwise positions and install the knobs.



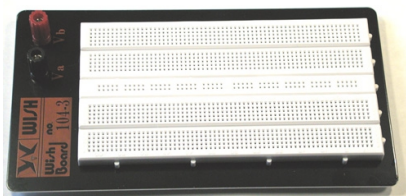
Done... Woo-hoo!

What Now?

Often, the first thing someone says is: "It sounds great, but how do I make it sound more: trebly, bassy, cooler, warmer, etc., etc.?"

First, experiment with the controls and gig with it for awhile to see what the pedal Can do. The Tremulous Bear is very versatile and a little idiosyncratic, and it will take you some playing to get to know it.

Having done that, Don't immediately start hacking into it and trying to change components. You put a lot of work into building this thing, and experimenting on a live pedal when you don't know where you want to wind up is a sure way to kill it. Buy another set of parts (even if you don't buy them from SBE), and set the circuit up on a solderless breadboard.



Or



You can find these on my Stock list and at many other mail-order sources; Radio Shack also carries them. Now you can change anything you want to quickly, and without soldering. Maybe you'll come up with a mod for your gigging pedal, or maybe you will wind up with a new design to build.

What component affects what aspect of the effect? This gets far more involved than I can possibly treat in this manual. The article on the hand-wired version includes a lot of detail about how the effect works that would be a good start. Also, there are many sources of information that you can consult, both print and on-line. Check the Links page of my web site for useful on-line resources. In particular, get to know the residents of the Forum at diystompboxes.com; the amount of accumulated knowledge and experience there is incredible.

I am available by e-mail at smallbearelec@ix.netcom.com if you have specific questions, especially about SBE's offerings. I do answer all technical inquiries, though I sometimes get backlogged, so please be patient.

It Doesn't (*&%\$?":!! Work!

First, take a shower and grab a bite, since we know that you haven't done either one since you started this thing; troubleshooting requires a clear head and normal blood-sugar level. The first rule thing to keep in mind is that projects like this are all-or-nothing. You went through dozens of steps in building it, right? If EVERYTHING is correct, the pedal works; if ONE thing is wrong, it doesn't. But you have something going for you: The instructions and photographs have been vetted by other builders, and you can rely on them. Use them as your bible, and you'll find out what's wrong.

First, define the symptoms:

- ◆ No sound at all with the stomp switch in either state? Or do you get straight guitar but no effect?
- ◆ No activity of the LED in either state of the stomp switch?
- ◆ The effect works, but the controls don't work right.

The most common mistakes in a build of this kind are:

- ◆ Putting a component in the wrong place.
- ◆ Orienting incorrectly a component that has polarity or a required orientation
- ◆ Mis-wiring to the external components
- ◆ Poor soldering/short circuit.
- ◆ Heat damage to a transistor or diode

First, review the components on your board against the pictures. Is **Every** resistor the right value and in the right position? Check off values against the parts list as you go. Do the same for the capacitors. Make sure that the right value pot is in each position.

Double-check the polarity of the electrolytic capacitors and the orientation of the diodes and transistors.

If the in-use LED is not wired correctly, it won't light. If this is the only problem you have, most likely either the LED is wired backwards or there is some problem in the driver circuit.

Go over the off-board connections. Remove the screws holding the board. Remove the knobs and retaining nuts from the controls and hardware from the stomp switch, and gently lift the whole works free. Verify that the off-board wiring conforms to the pictures and text. **Every wire must go from where the instructions tell you to the correct destination, and every solder joint must be solid.** Turn the PC board to the solder side and make sure that every joint is solid. Be sure that there are no broken traces or a blob of solder causing a short. A magnifier and a strong light are helpful when doing this. **99%+ of problems with kits of this kind and level relate to poor soldering!**

While you have the guts out of the case, try re-connecting the battery, guitar and amp and see if the pedal works. If it does, maybe something was shorting. Try reassembling and see if the pedal still works.

If you still don't get the effect with the guts out of the case, you may need to buy or borrow a multimeter and take a few voltage measurements. If you have never done this, try to get some local help from someone who has. If you have someone helping you, that person should help you make these measurements and write down the results.

First, measure the battery voltage with the battery disconnected from the pedal. It should be at least 9.0 volts, and will more likely be higher if the battery is fresh. Re-connect the battery, and be sure that the voltage on the Cathode (bar side) of D2 is within 200 millivolts or so of the no-load battery voltage. If that's not right, check Everything related to power!

Get the LFO (low-frequency oscillator) working first. Make sure that the 555 is seeing supply voltage at pins 4 and 8 and ground at pin 1. With the speed pots at minimum, you should see the output voltage of the 555 (pin 3) go up and down between zero and about 8.2 volts. Should be the case in either state of the stomp switch. If this is not happening, look for wiring problems related to the 555.

In order for the LED to pulse, the LFO (low-frequency oscillator) must be able to turn on. For this to happen, the Base of transistor Q3 must be able to switch between ground and about .7 volts when the stomp switch is clicked. If this is not happening, check the stomp switch wiring and related traces on the board.

If there are problems with the sound, first check the supply voltages on the modulator side. You need to have +8.5 volts at pin 8 of the op-amp chip and +4.25 on pin 3 and pin 5. If the voltages are there, see if you can narrow the trouble to one stage by using the guitar amp as a signal tracer. Again, you may have to test continuity of each PC board trace and connection with an ohmmeter.

If you get stuck, e-mail me at smallbearelec@ix.netcom.com. I have been through troubleshooting the Trem numerous times and have always been able to work through whatever the problem is. You may also be able to get some help on one of the effects forums if you are patient and polite.

The parts list that follows identifies all of the components by the designations that appear on the printed circuit board.

Parts List

Quantity	Description	Designation
	Resistors – ¼ Watt, Carbon Film, 5% Tolerance	
3	22K	R2, R3, R10
1	47K	R5
1	33K	R6
1	2.7K	R7
1	180K	R9
1	100 Meg	R12
1	3.3K	R13
1	3.3K to 10K	R14
1	100 ohms	R15
6	100K	R16, R18 R19-R22
1	1 Meg	R17
1	150K	R23
1	5.6K	R24
	Potentiometers – Alpha 16mm PC Mount	
2	500K Reverse Audio	R1, R4
1	1 Meg Reverse Audio	R8
1	10K Linear	R11
1	50K Audio	R25
	Capacitors – Polyester Film	
1	.1 mf.	C3
1	.22 mf.	C4
1	.01 mf.	C8
1	.68 mf.	C9
	Capacitors – Electrolytic, Radial, 16 Volts	
2	100 mf.	C1, C6
2	10 mf.	C2, C5
1	2.2 mf. (tantalum)	C7
	Diodes	
2	1N914	D1, D2
1	1N5818	D3
1	5mm Red High-brightness LED	LED1
	Transistors	
1	2N7000	Q1
1	2N2222A	Q2
1	MPSA14	Q3
	ICs	
1	TL072	IC1
1	TS555	IC2
	Switches	
2	SPDT Toggle	S1, S2

1	DPDT Alternate Action – Alpha SF12020	S3
	Jacks	
1	Open-Circuit 1/4” Stereo	J1
1	Open-Circuit 1/4” Mono	J2
1	External Power	J3
1	Photocoupler	LDR1/LED2
	Case, Fittings, Hardware	
1	“BB-Size” Enclosure With Four Screws	
1	PC Board	
2	IC Socket	
3	Rubber Feet SJ-5023	
1	5mm LED Mount	
1	9-Volt Battery Drawer	
5	Knobs, Fluted, Miniature	
1	Decal	
1	Hookup Wire – 12” Black, 12” Red, 6” Yellow, 6” Blue, 6” Orange, 6” Green, 6” Blue	