UVICS

The circuit of the

UniVibe In

a

Crybaby Shell

Includes

Son of UVICS

Updated 6-3-2013

Table of Contents

Introduction: What's a UVICS and how is it doing all those neato things?

I released the first DIY layout for a Univibe clone to the internet back in the 1990s, back when the only sharing medium was usenet. This layout went through a number of refinements, eventually becoming the Neovibe of today's DIY effects world. There have been a huge number of these things built. I don't have any way to find an accurate count, but I suspect that there have been thousands, just from the volume of email I have received asking for technical support in one form or another.

The original layout that would become the Neovibe of today was deliberately compact; early on I decided it would be a good thing to mount inside a Crybaby wah shell. The original did in fact fit inside the shell, but it was not specially set up to fit the molded-in standoffs of the wah shell. Several people built them into wah shells, and I always intended to redesign the physical layout of the Neovibe to fit a wah shell directly. I finally got around to doing this.

UVICS? Why such a silly name? UVICS stands for UniVibe In a Crybaby Shell. I originally called it a "Cry-Vibey". This resulted in a flame-storm from a small effects maker who made and sold a proprietary product; the biggest beef was that yet a third person had called their version of a Univibe circuit squashed into a wah shell a "Cry Vibe". It was an odd situation with one person crying foul over a name that was supposedly a trademark that belonged to someone else who was not selling any products. Bizarre.

But as the bard said, would it smell less sweet if it was called something else? Nah. So it's a UniVibe In a Crybaby Shell. UVICS. Nice ring to it, I think.

But - *Son of UVICS?*

Why not? UVICS offered the original circuit of the 1960s Univibe in a wah-shell-friendly format. But the DIY community being what it is, there were requests for various mods that have come up over time. So I went back to the layout and tried to see what I could stuff into the limited space.

UVICS offered some of these, including the biggest things, a single-section speed pot and a 9Vdc power option, as a separate small PCB that could be placed at the front of a wah pedal. But it would be really cool to get them all on the same board right?

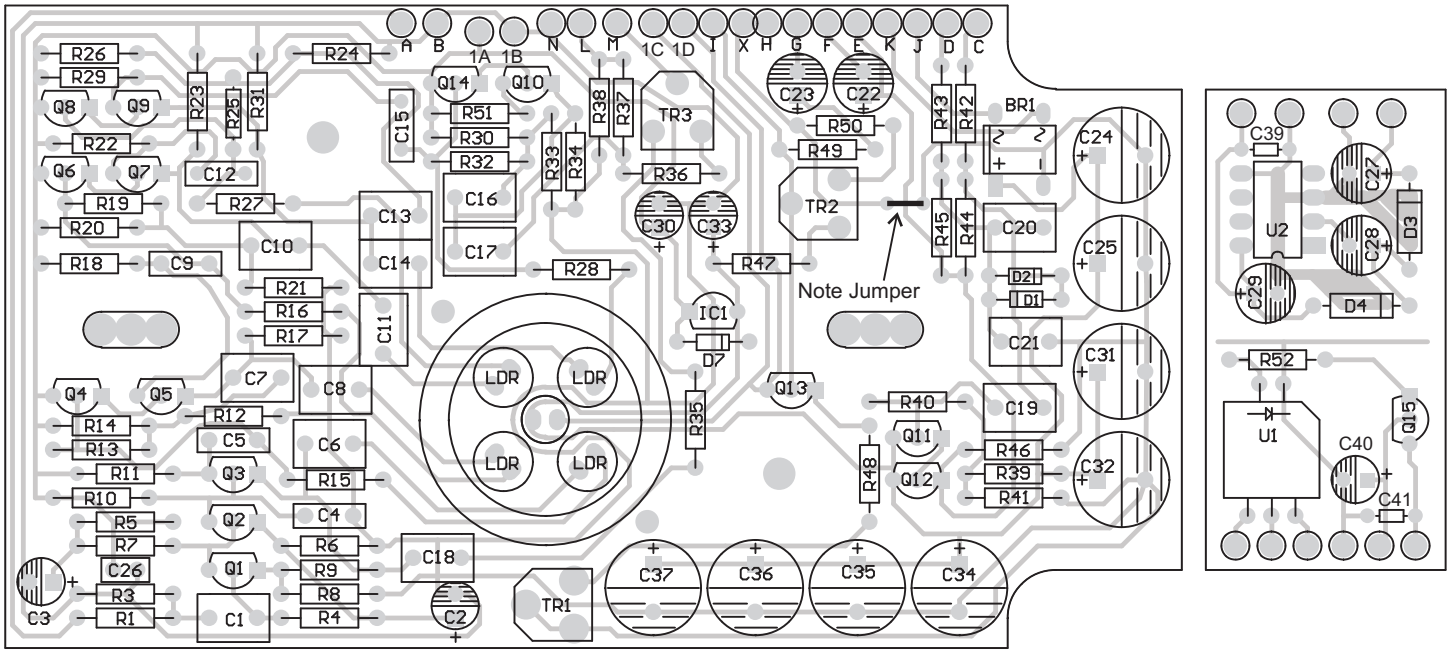
By standing the resistors on end, I got enough PCB space to put a number of mods onto the single PCB that became Son of UVICS. For the DIY tinkerer, it's an easy way to get a highly customized 'vibe in a wah pedal. It's a little more difficult to get parts placed and soldered in, but if you want to do it, it's there.

Mods!~! What Mods? The original Univibe was a good fit for what it did, and became justly famous for it. But today's effect user is much more sophisticated and demanding than the effects user of the 1960s. Effects need to not only do good sounds, but they have to play well with other effects, and not have inadvertent side effects on the guitar's signal. So some facets of the original Univibe design have come to be viewed as needing changed or updated. For more information along these lines, see "The Technology of the Univibe" at http://www.geofex.com. The most widely requested mods include:

- "unity gain"
- true bypass
- higher input impedance
- phaser depth/balance trim
- effect indicator blinking with the LFO

It occurred to me that it would be really useful to include a the ability to use the existing wah pot in the wah shell; I came up with a way to do this a long time ago. I'm told it has been independently reinvented. A DIYer also tried running a Neovibe from a charge

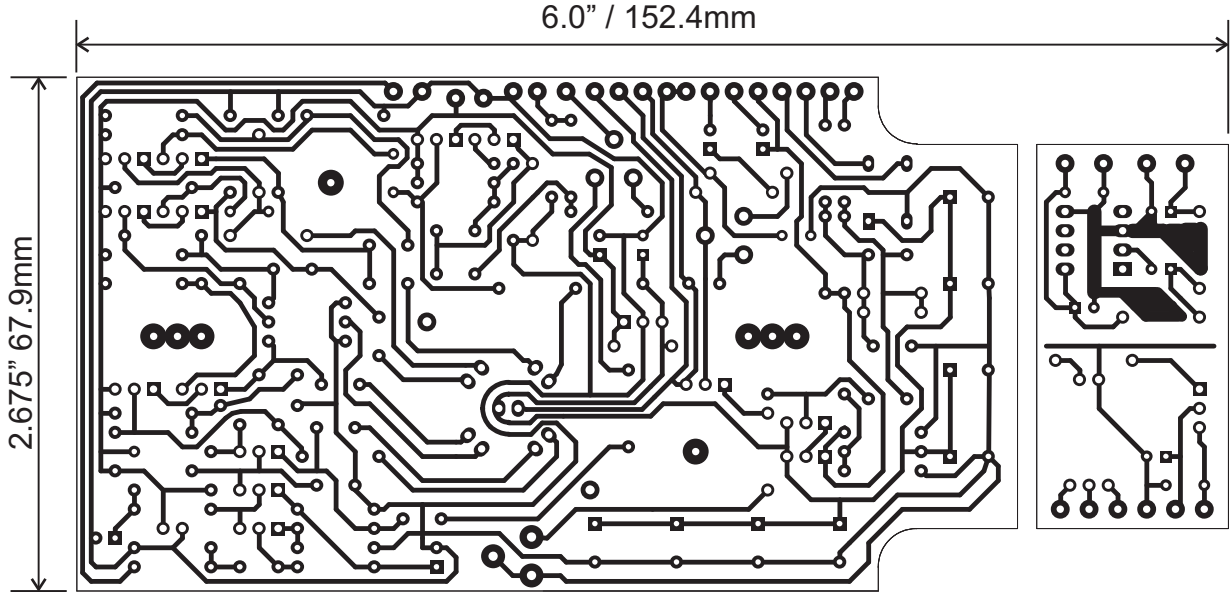
UVICS-3



Above is the parts placement and “transparent traces” as seen from the component side of the PCB.

Below is the actual size PCB pattern, suitable for printing for direct toner transfer, along with the actual size of the board. Print this on paper and measure the size of the you get to ensure it comes out the correct size. When it’s right on paper, print on your transfer medium.

The “baby board” on the right side of the main PCB is intended to do a couple of things that the vintage circuit does not do, and to make it work inside a wah shell on a modern pedalboard more easily. You can either (a) not build the baby board at all if you don’t need 9Vdc to 17Vdc power and reusing the single existing wah pot instead of manually fitting a dual pot into the wah shell. You can also cut the baby board in half across that line in the middle and use either half, or place the two smaller boards into other places in your wah shell.



pump converter to make 17Vdc from 9Vdc. It seems to work OK, so I included this as an option as well.

For people who want the original styled AC power, there is space on the PCB for the original rectifiers and filters to make DC power out of an AC source.

But but but... do i need UVICS or Son of UVICS? That depends on a number of things. Probably the easiest way to tell is to ask yourself what functions you absolutely, positively have to have, and how much you're willing to do to get them.

1. Do you need switchable phase capacitors or are the original Univibe cap values OK? If you have to have switchable values, you have to go with Son of UVICS.

2. Can you stand resistors on end and build a fairly crowded PCB? If not, you need UVICS, not Son of.

3. Do you demand every possible mod? Or do you want the easiest build for a Univibe sound? Son of has more mods available if you'll use them. UVICS is an easier build, IMHO.

Both boards are laid out so that if you can do toner transfers with 0.025" traces and 0.010" spaces between conductors, they will be reasonably easy to do a board from.

What happened to UVICS-1 and -2? Is there a UVICS 4? UVICS-1 and -2 were steps along the way. I had a very brave soul who volunteered to build one as a test. he built the -1. It worked fine, but there were some refinements suggested. Those got rolled into UVICS-2 and finally UVICS-3.

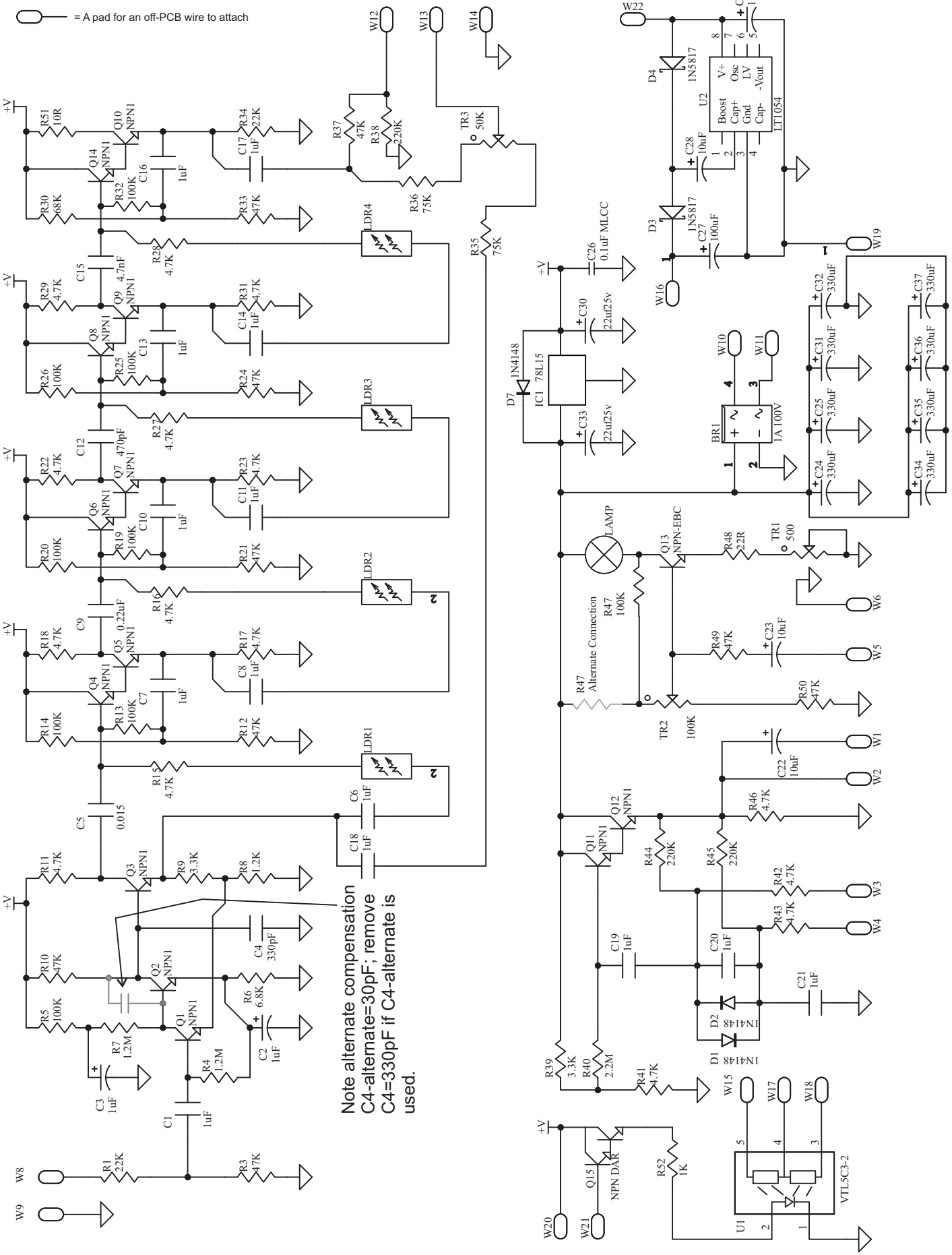
There is a UVICS-4. Shhhh. It's a secret. It adds to the UVICS-3 the ability to use the rocker to cancel the effect the same way the original Univibe did. Frankly, I think that most people won't like that once they do it. If you have a real, screaming need to do "cancel" instead of bypass, get in touch.

Son Of UVICS is a step a little bit sideways from the 1,2 3, and 4. I did it as a personal challenge to me to put all that I could on a single board fitted to the wah shell, and to do it with all the mods I could stuff onto it. I normally have a set of PCB layout rules I use that result in things being easy for DIYers to make and populate the PCBs. For Son Of, I relaxed those just a little to stuff more mod space onto the PCB.

Updates:

6-3-2013: I had a series of questions from a fellow who built a UVICS, and wound up getting the board to tinker with for a few days. There were some minor problems with wiring and soldering, but there were three problems that deserve mention here. These are (1) there is a jumper that the board means that was not mentioned in this write-up, my bad. I've fixed that in this version of write-up; (2) the compensation scheme with C4=330pF to ground as in the original Univibe did not prevent oscillation of the first three-transistor compound. However, changing to a 30pF capacitor between Q2 collector and base, the modified compensation I mentioned in some posts on the net, did work fine. I'll show how to do that; and (3) the charge pump power supply the fellow used resulted in power that was ugly and noisy as sin itself. This was not the charge pump PCB I laid out for this project, but does indicate that you need to take care with this. I'll mess with this some more and see if I can't make the thing more noise immune.

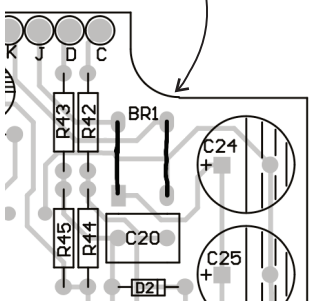
UVICS-3 Base Schematic

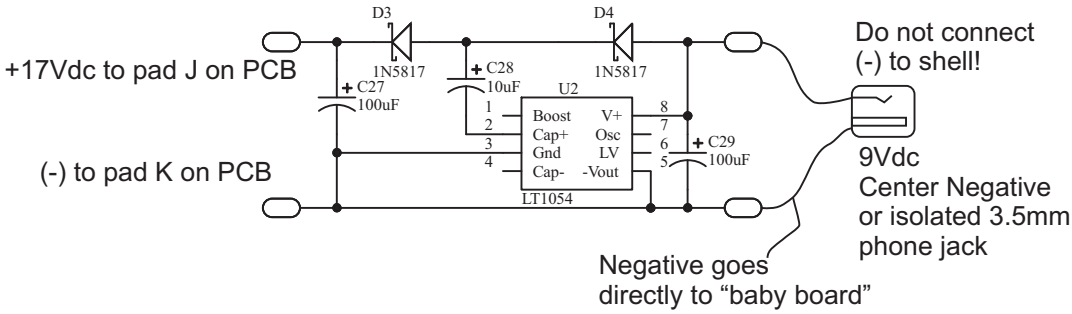


UVICS-3 Additions and Modifications Using the Baby Board

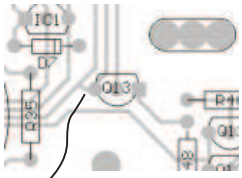
Run UVICS from +9Vdc adapter

Do not use Br1, and short across its pins like this:





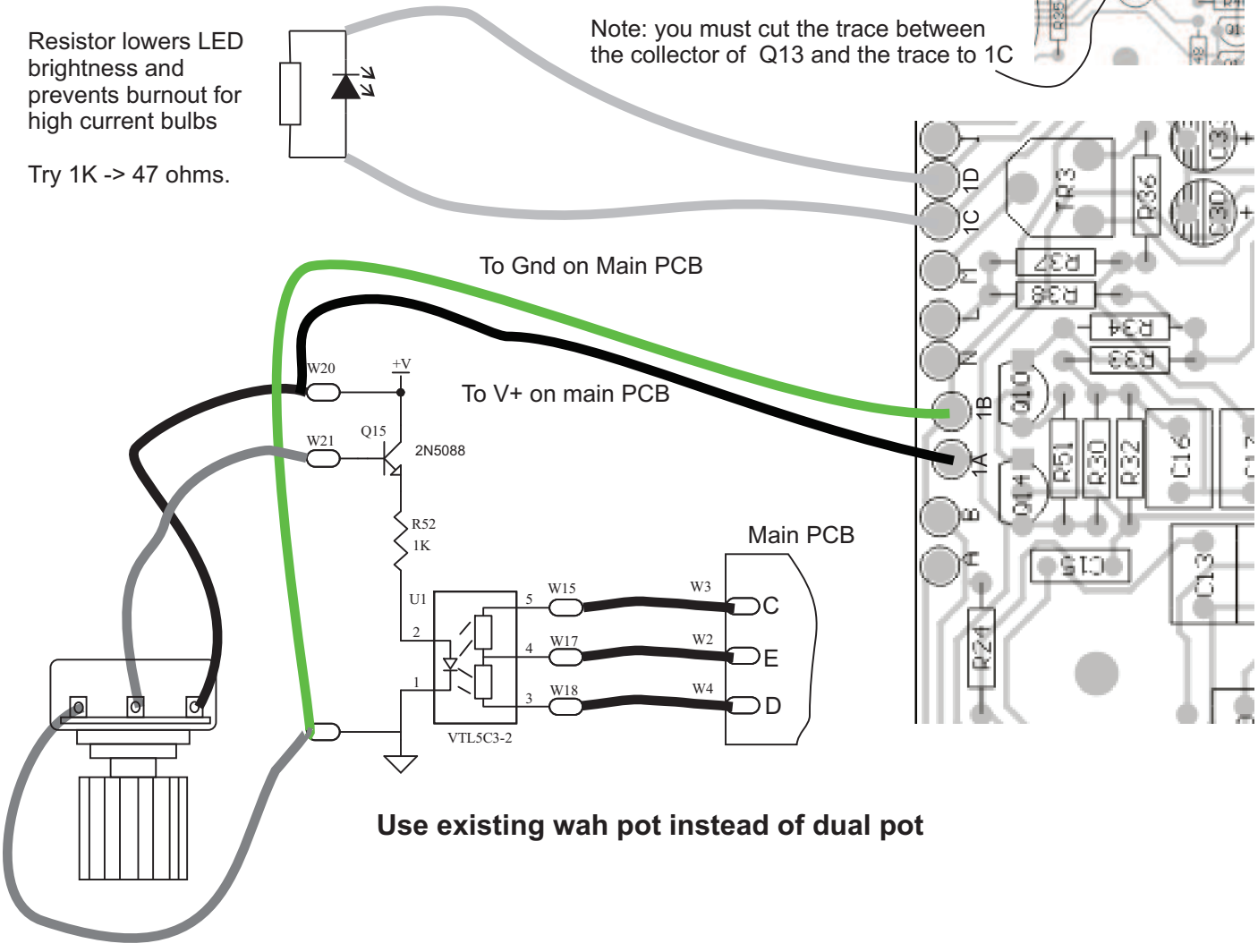
LED following LFO



Note: you must cut the trace between the collector of Q13 and the trace to 1C

Resistor lowers LED brightness and prevents burnout for high current bulbs

Try 1K -> 47 ohms.



Use existing wah pot instead of dual pot

UVICS-3 BOM

Basic Parts on main PCB

Value	Designator	Note/Description
0.1uF ceramic	C26	MLCC Epoxy dipped
330pF	C4	Ceramic type COG or NPO
470pF	C12	Ceramic type COG or NPO
4.7nF	C15	Box film
0.015uF	C5	Box film
0.22uF	C9	Box film
1uF NP	C1, 6, 7, 8, 10, 11, 13, 14, 16, 17, 18, 19, 20, 21	Box film, 1uF NP, or 1uF polar
1uF Electr	C2,3	Aluminum Electrolytic
10uF	C22,23	Aluminum Electrolytic
22uf25v	C30,33	Aluminum Electrolytic
330uF	C24,25,31,32,...	Al Electro; the size is important - get 10mm/0.4” diameter caps less than 13mm long. Wah shells vary, and caps are tall, so be prepared to tinker. See below.
10R	R51	1/4W carbon film- 0.4”lead spacing
22R	R48	1/4W carbon film- 0.4”lead spacing
1.2K	R8	1/4W carbon film- 0.4”lead spacing
3.3K	R9,39	1/4W carbon film- 0.4”lead spacing
4.7K	R11, 15, 16, 17, 18, 22, 23, 27, 28, 29, 31, 41, 42, 43, 46	1/4W carbon film- 0.4”lead spacing
6.8K	R6	1/4W carbon film- 0.4”lead spacing
22K	R1, 2, 34	1/4W carbon film- 0.4”lead spacing
47K	R3, R10, 12, 21, 24, 33, 37, 49, 50	1/4W carbon film- 0.4”lead spacing
68K	R30	1/4W carbon film- 0.4”lead spacing
75K	R35, 36	1/4W carbon film- 0.4”lead spacing
100K	R5, 13, 14, 19, 20, 25, 26, 32, 47	1/4W carbon film- 0.4”lead spacing
220K	R38, 44, 45	1/4W carbon film- 0.4”lead spacing
1.2M	R4, 7	1/4W carbon film- 0.4”lead spacing
2.2M	R40	1/4W carbon film- 0.4”lead spacing
500R	TR1	Trimpot; Bourns type 3306
100K	TR2	Trimpot; Bourns type 3306
50K	TR3	Trimpot; Bourns type 3306
DISK LDR	LDR1...LDR4	
LAMP	DS1	12V/25ma or similar
NPN	Q1-Q12, Q14	2N3906, 2N5088, etc.
NPN-Darlington	Q13,	1W TO-92 package; example: MPSW45
1A 100V	BR-1,	1A or more 100V or more diode bridge in DIP-6 package
1N4148	D1,2,7	signal diode
78L15	IC1	voltage regulator in TO-92
Dual 100K-250K rocker pot with gear		this will be one of the harder ones to find and fit, most likely
50K linear	Depth Pot	
SPDT	Chorus/Vibrato Switch	
Wah shell, with jacks and SPDT switch		

Optional parts for baby board power by 9Vdc:

LT1054	U2	charge pump converter IC
1N5817	D3,4	Schottky diode, 1A
100uF	C27	Aluminum electro
10uF	C28	Solid Tantalum capacitor: WATCH THE POLARITY CAREFULLY
10uF	C29	Solid Tantalum capacitor: WATCH THE POLARITY CAREFULLY
0.1uF MLCC	C39	MLCC ceramic capacitor

Optional parts for using the single wah pot

2N5088	Q15	NPN high gain, EBC pinout
1K	R52	current scaling resistor; adjust to fit your transistor and LED/LDR
VTLC3/2	U1	LED/LDR with center tap VTLC2/2 may work too.
1uF	C40	Aluminum electro
0.1uF MLCC	C41	MLCC ceramic capacitor

This list is for the stock parts. Mods may change or extend the list.
- For C24/25/32..36, use a total capacitance which gives you appropriately little ripple, not necessarily to fill up all the capacitor positions. Select 10mm diameter, less than 13mm high. I picked 330uF/35V in case of a high AC voltage, but if you use a power supply which produces even less maximum voltage - say, 18-22Vdc at the caps - then you can use lower voltage but higher capacitance caps of the same physical size. There are 470uF/25V caps at Mouser that fit the 10mm wide by 12mm tall spacing. Another possibility is to get 1000uF capacitors with a diameter less than 12mm, but longer. These can be laid down in the space provided for the stand-up capacitors.
- If you can't get short enough caps, the further from the heel you go the more height you have, so the positions along the bottom are some help.
- If you an AC-output power adapter, you will need a lot of capacitance; 2000uF should be enough, and that's six 330uF caps. For an AC supply, be sure the incoming AC on the power jack does NOT contact the grounded shell.
- If you have a DC output supply that's not filtered, you still need a lot of capacitance. In this case, use lots of capacitors and put two jumpers between the pads for the bridge rectifier to let the DC directly through to the capacitors. A bridge rectifier just gets in your way if you have DC coming in already.
- If the DC supply is rectified and filtered, but not regulated, you may need less total amount of capacitance. If the supply is DC, filtered, and regulated, you'll only need one or two caps. Same comment about bridges.
- The signal capacitors are almost all laid out for 0.2" lead spacing 50V box style film caps. You can adapt other caps, but watch the sizes. The 0.22uF phase cap *is* available as a box cap less than 11mm tall. I have several hundred for other reasons.
- Remember to put in the jumper shown on the parts location diagram.

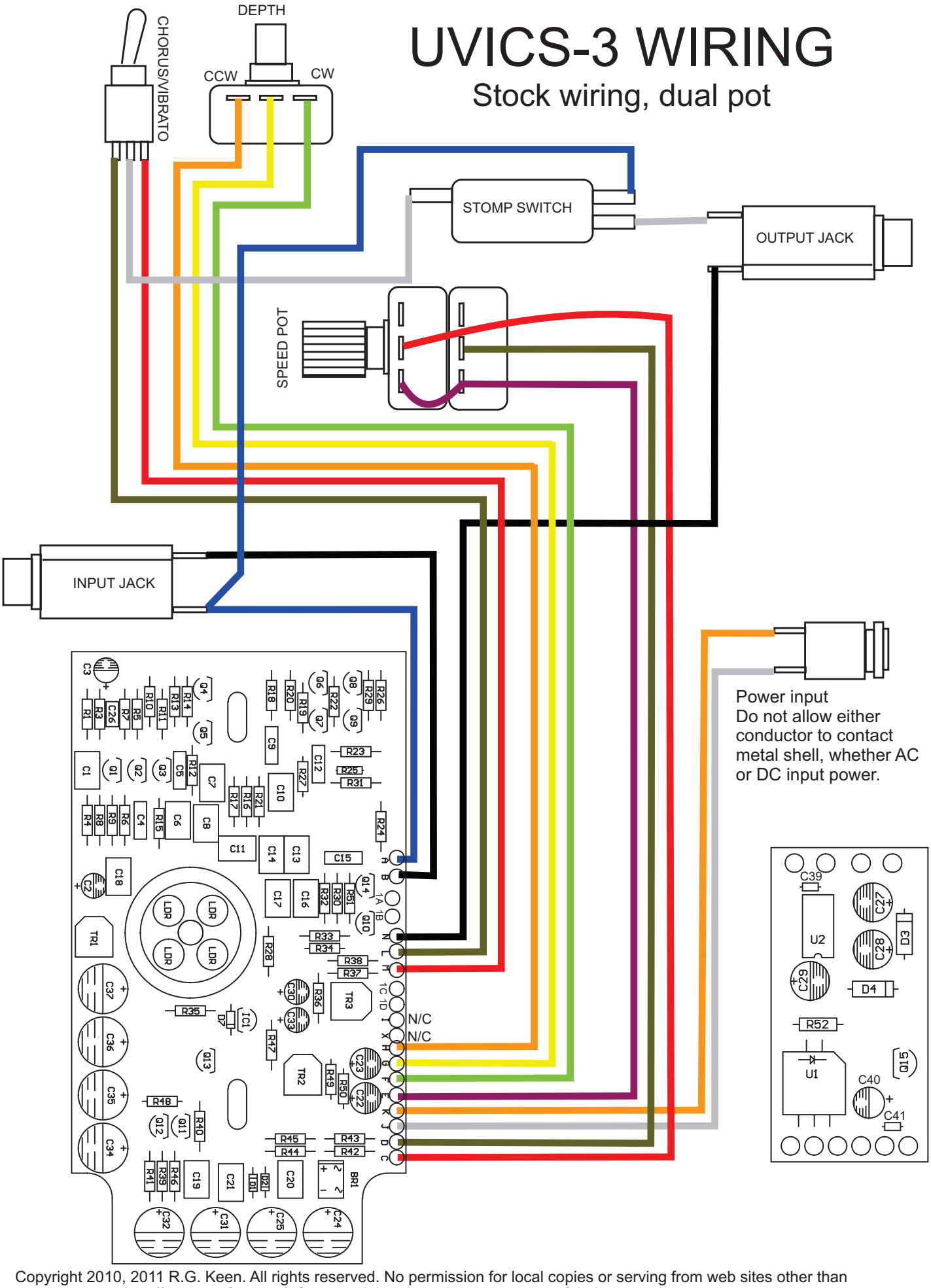
The board itself lets you have a number of options.
- You can raise the input resistance to over 1M by changing the values of the first two resistors.
- There are the stock two inputs from the original univibe; you won't need both for use in a crybaby shell, so leave off the input resistor for either A or A1 inputs.
- Likewise, you probably won't need the "cancel" function, but the pads are there to do it if you want.
- You can go to darlingtons by shorting the base pad to emitter pads on the phase array transistors and the Q11/Q12 positions in the LFO.
- You can hack on a "stereo" output if you like because I put in the collector resistor position and second transistor position for the fourth phase stage position. You'll have to make a separate baby board to put on the stereo mixing, and I didn't think many people would do it, so I didn't put it on the PCB where it would have had to go.
- The 1uF film capacitors I used so profusely don't have to be film. They were polarized electros in the original. You can use polarized electros, or you can use NP electros, or you can use film. The board lets you put them in.

A DIY project is a bundle of raw potential, a chance to create something new.

Some tips on fitting the PCB into the shell:
- The mounting holes need to be about 0.137"
- The reference locations are the two single mounting pads which fit over the raised bosses in the shell. The other two three-in-a-row locations are for drilling three holes then cutting/filing the holes together into a slot to fit over the threaded ends of the tension adjuster screws. I put the center hole over the locations of my adjuster screws, but I understand that shells vary in the adjuster locations, so I put extra space there to make a slot for it to fit more shells.
- The tightest height restraint is at the heel end where the power filter caps are. The caps can't be any taller than about 13mm or they'll hit the cover. To avoid this, you want capacitors which are 10mm/0.4" diameter, with 5mm/0.2" lead spacing, and less than 13mm high. Mouser's part selection app lets you pick by size among other things, which is very handy. I did the search for parts before I did the layout, and they stock several 330uF/35V capacitors with a diameter of 10mm and height of 12mm. That was the reference cap I used.
- The light shield may need some ingenuity. Of course, anything that keeps light off it while it operates will work. In fact, just the shell itself, no light shield, will probably work fine in actual use. But for debugging with it open, you may need something; and an internally reflective shield does the best job of "mixing"the light so the LDRs are illuminated evenly. You can even tape down the aluminized baggie from potato chips over them. It does not need to be perfectly shaped and sized, just there. Aluminized cardboard works, as does thin metal folded into a box, whatever fits. My recent tinkering turned up PVC pipe. The nominal 3/4" pipe can be sliced into a short ring of less than 16-17mm height (that's about what you have at the heel end of the light shield) and glued onto the PCB. Then you can tape/glue/tie a reflective top onto it. Finding stuff that happens to work is one of the joys of DIY. The world is practically **full** of things that happen to be shaped right for a purpose that wasn't what they were intended for. I give myelf extra points when I spot something that can be used for an application the original designer would never have thought of when they made it.

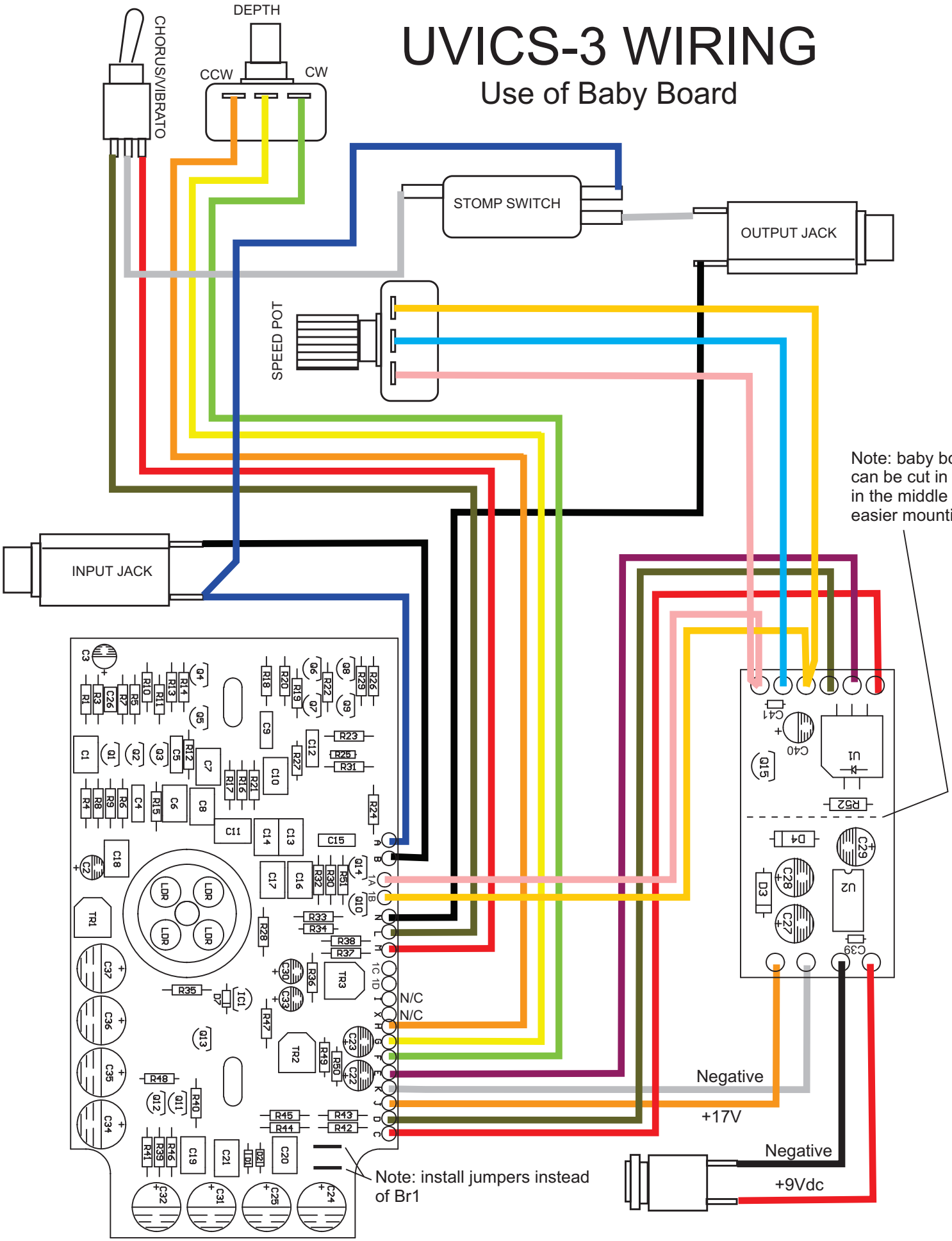
UVICS-3 WIRING

Stock wiring, dual pot



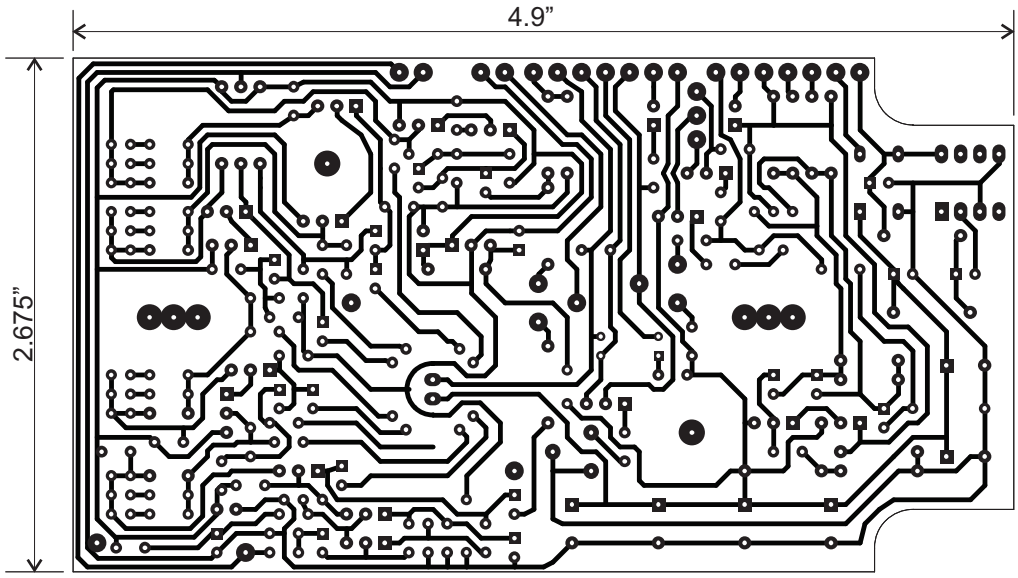
UVICS-3 WIRING

Use of Baby Board

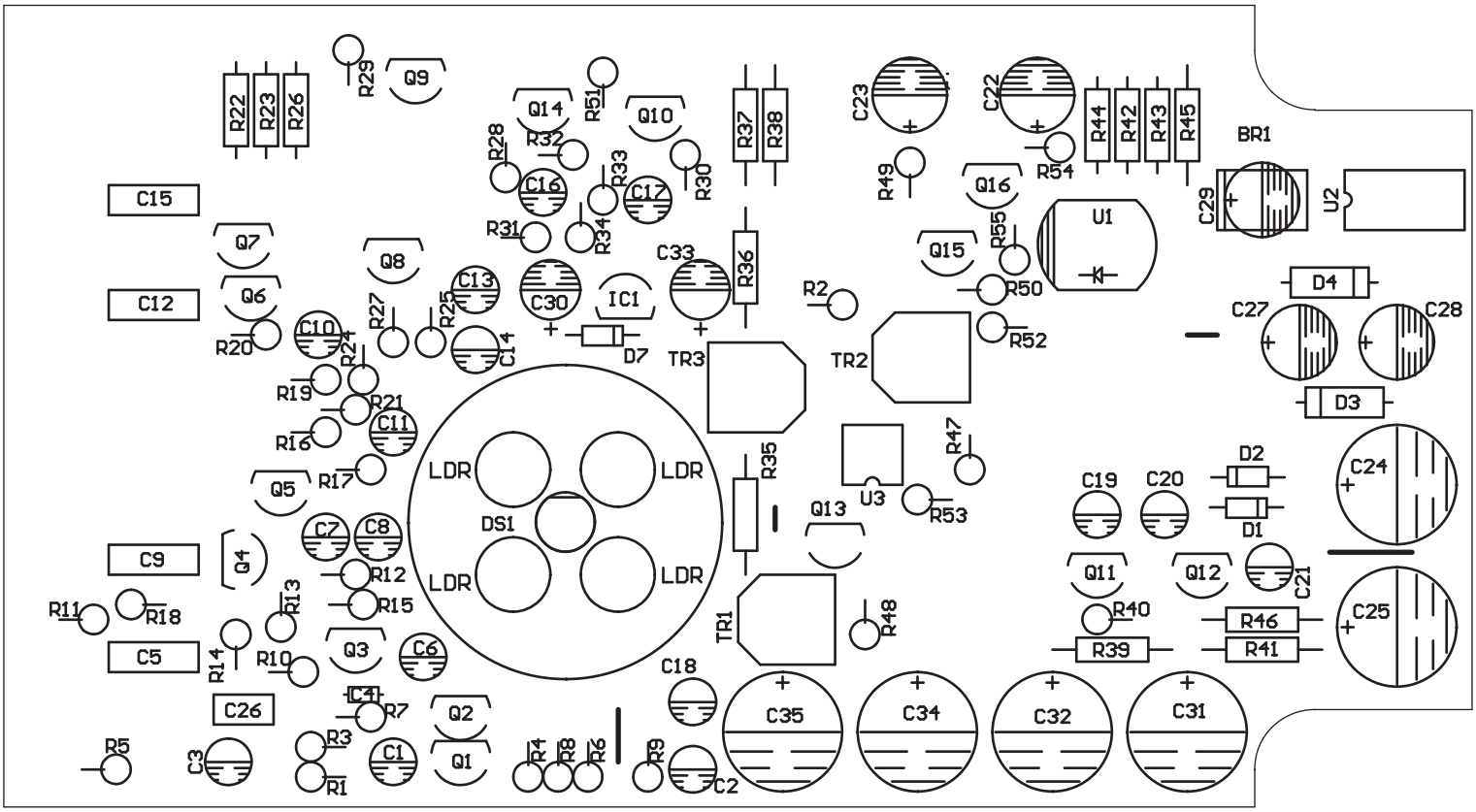


Son Of UVICS

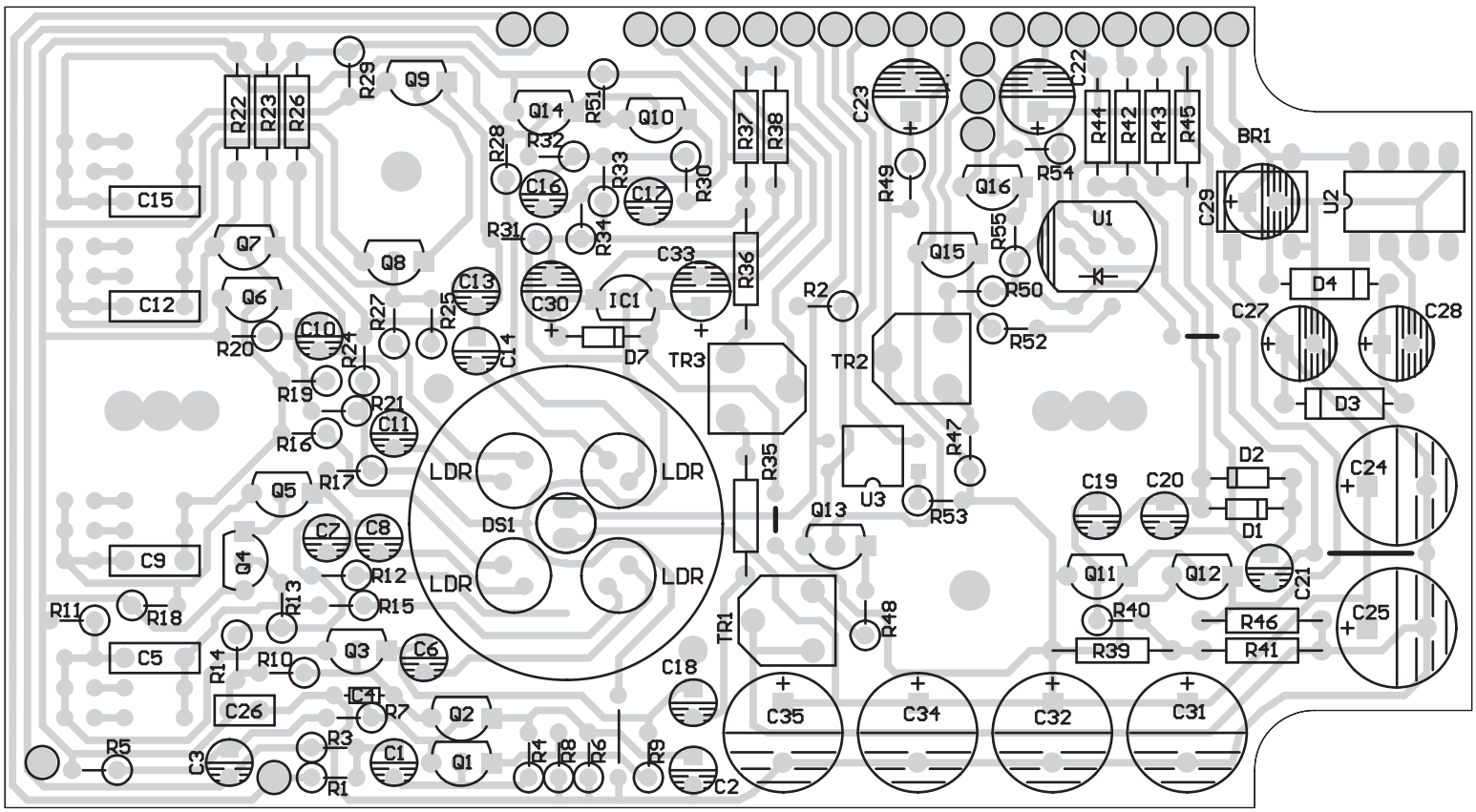
Actual Size Toner Transfer Image



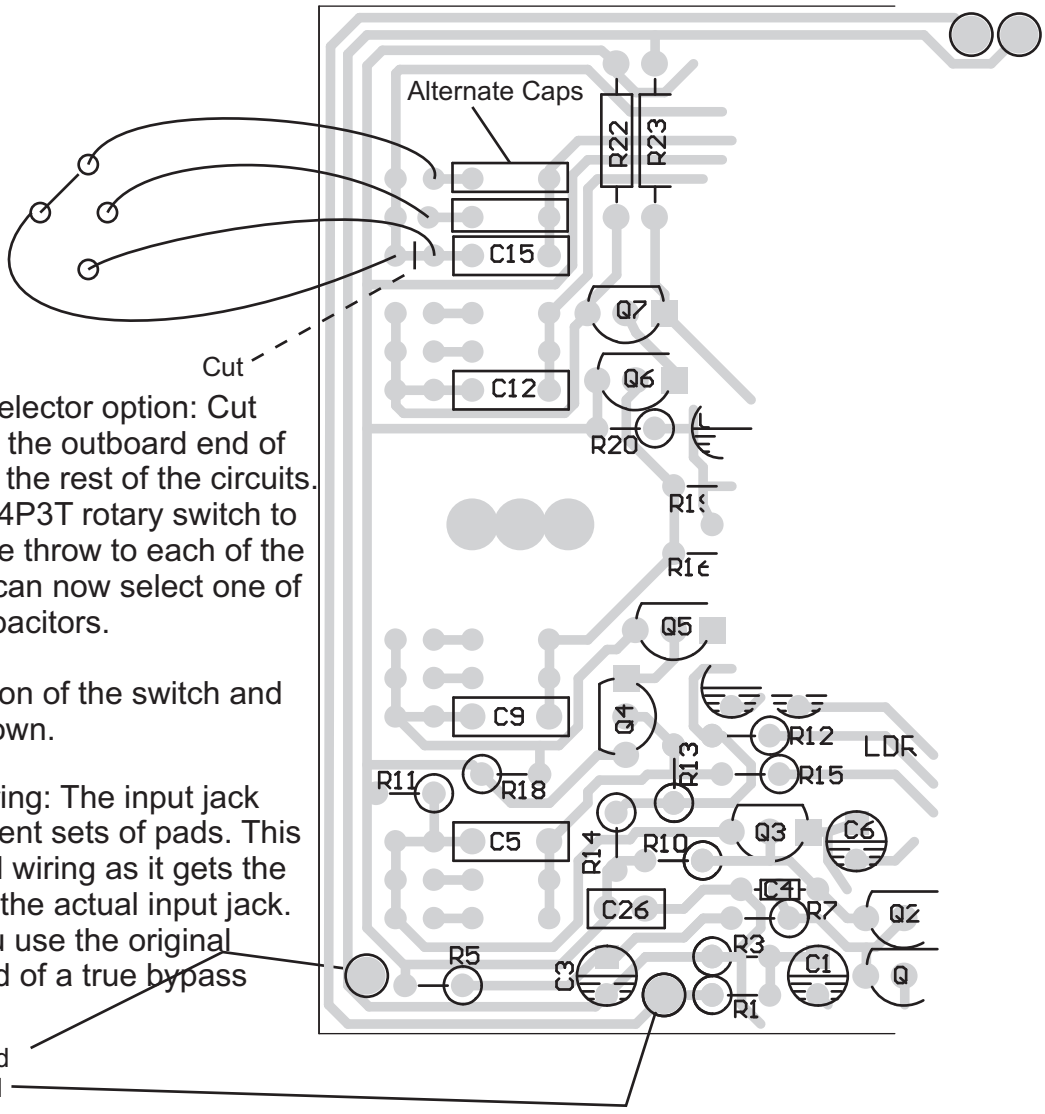
Component Location Diagram



Board Preparation



Selective Population Options for Son of UVICS (1)

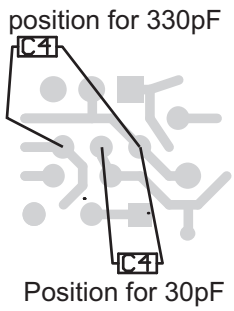


1.Three- way capacitor selector option: Cut the trace which connects the outboard end of C5, C9, C12, and C15 to the rest of the circuits. Then wire the poles of a 4P3T rotary switch to the common line, and one throw to each of the capacitor positions. You can now select one of three different sets of capacitors.

For clarity, only one section of the switch and optional capacitors is shown.

2. Alternate Input jack wiring: The input jack can be wired to two different sets of pads. This may simplify your internal wiring as it gets the pads for the input nearer the actual input jack. This may be helpful if you use the original "cancel" operation instead of a true bypass switching arrangement.

3. Alternate compensation: The original circuit was compensated with a single 330pF capacitor to ground at C4. With modern transistors, the circuit is actually more stable if C4 is connected between base and collector of Q2 and reduced to 30pF. On the layout, C4 has three holes. The outer two holes are the original circuit position. If you instead use a 30pF in the right most two holes, you get the newer compensation.

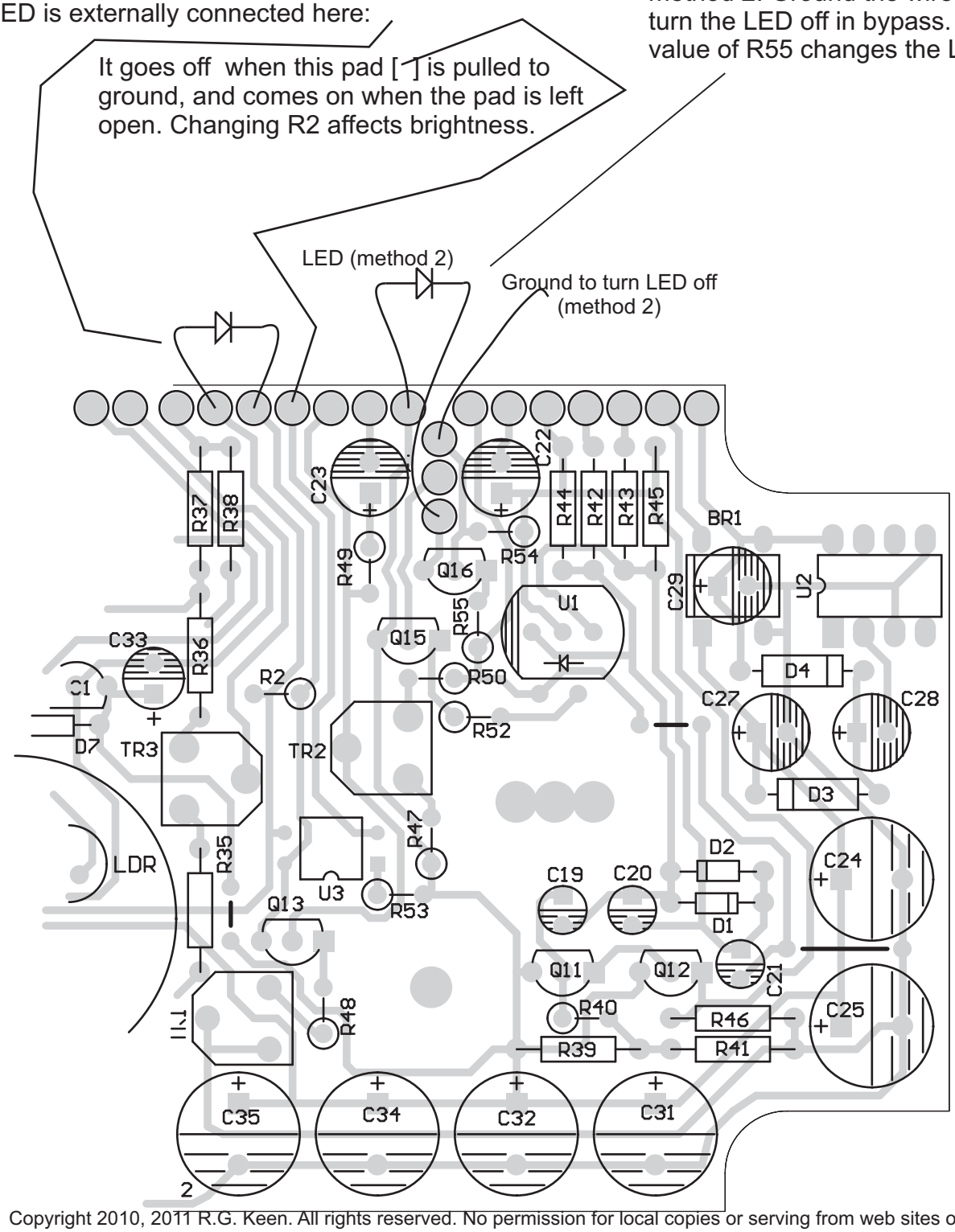


Selective Population Options (2)

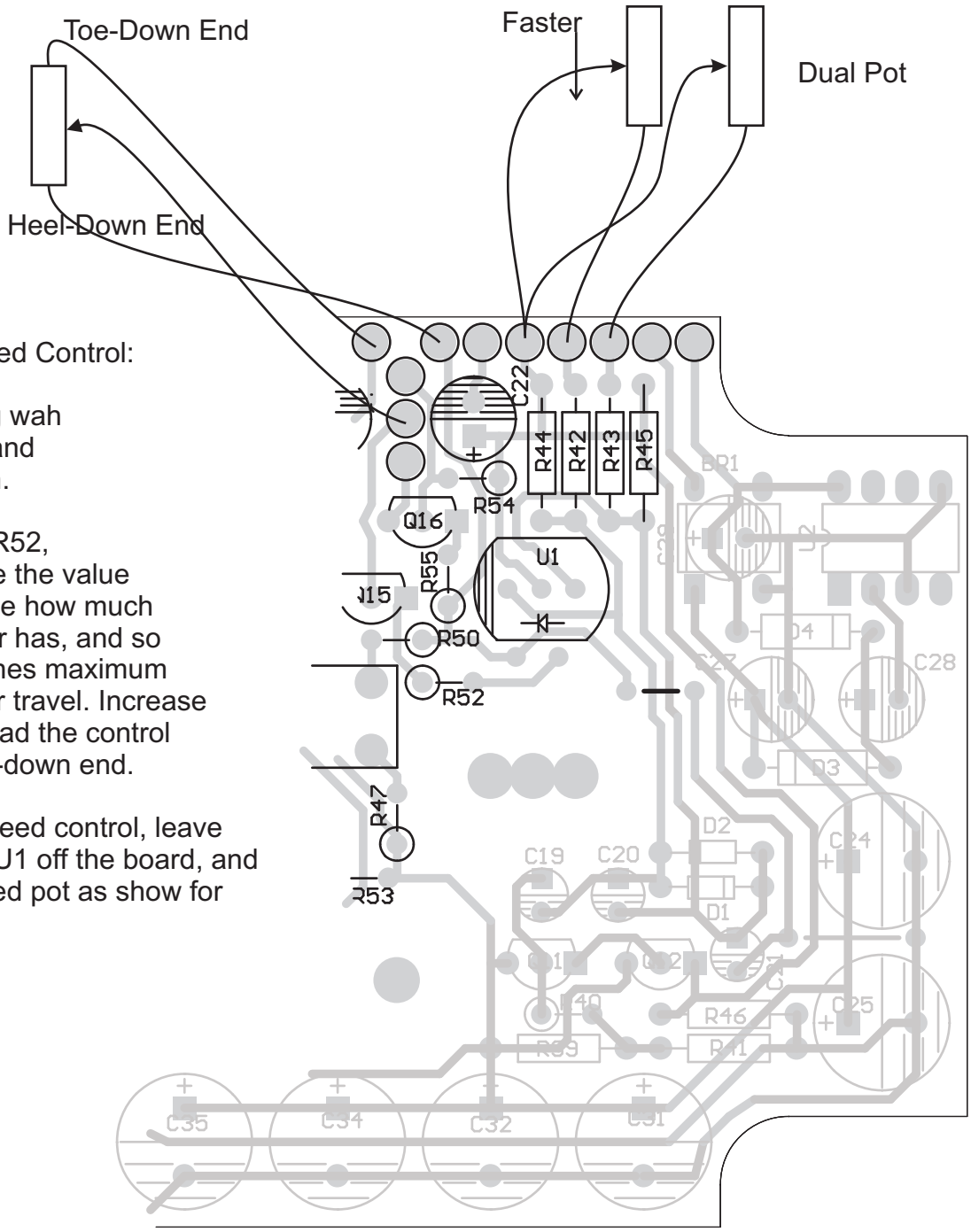
Blinkin' Lights: There are two ways on the PCB to make a blinking indicator light that blinks in time with the LFO, but goes off when the pedal is bypassed/canceled.
- Method 1: LED in series with the bulb. For this method, populate R2, R53, and U3, and leave off R54, R55, and Q16.
The LED is externally connected here:

It goes off when this pad [ ] is pulled to ground, and comes on when the pad is left open. Changing R2 affects brightness.

- Method 2: LED driven by a transistor from the LFO output. For this method, Leave off R2, R53, and U3, but populate R54, R54, and U3. Connect the LED as shown for method 2. Ground the wire as shown to turn the LED off in bypass. Changing the value of R55 changes the LED brightness.



Selective Population Options (3)



Single Pot Speed Control:

Use the existing wah shell 100K pot and wire it as shown.

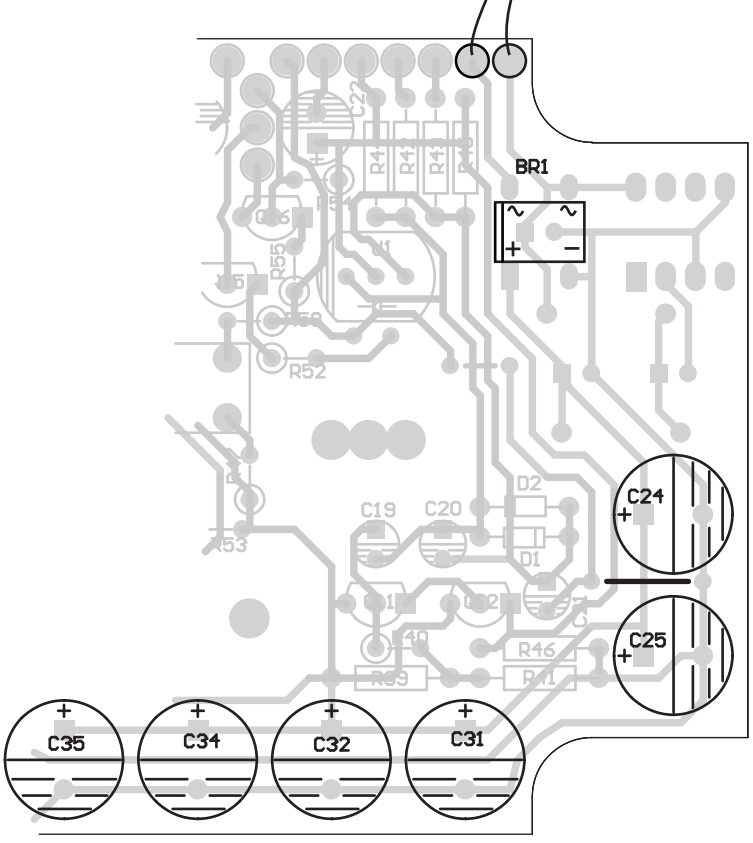
Solder in Q15, R52, and U1. Change the value of R52 to change how much “gain” the rocker has, and so how fast it reaches maximum speed for rocker travel. Increase its value to spread the control towards the toe-down end.

For dual-pot speed control, leave Q15, R52, and U1 off the board, and wire a dual speed pot as show for “Dual Pot”

Selective Population Options (4)

AC wall wart power supply: Bring in 14Vac to 18Vac from AC-output wall wart. DO NOT allow either wire to be connected to signal ground.

Populate BR1 and C24, 25, 31, 32, 33, 34, and 35.



9Vdc input option: leave off BR1 and all but two of C24, 25, 31, 32, 33, 34, 35. Populate C27, 28, 29, D3, D4, and U2. Add wire jumper as shown around C29.