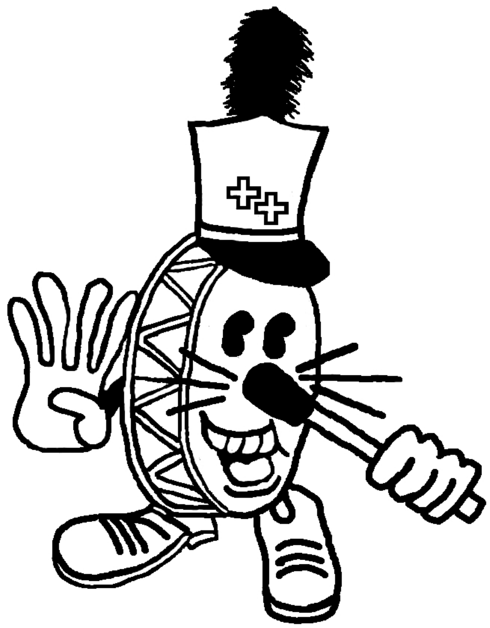
The Quad Bass++ Drum Voice



Designed By Thomas Henry

Quad Version Distribution by sMs Audio Electronics

83 Cranberry Circle, Medford NY 11763
Sales: thex@optonline.net

sMs Audio Electronics
DIGITAL & ANALOG
MUSIC PRODUCTS

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

Preface:

Some words by Scott Stites used with permission from the pages of “Birth of a Synth” DIY website: http://mypeoplepc.com/members/scottnoanh/birthofasynth/index.html

At this writing (August of 2008) the Bass++ is the latest of a triumvirate of percussive voices designed by Thomas Henry since the halcyon days of Midwest Analog.

The name "Bass++" is derived from the fact that this module can deliver a supremely fine bass voice. The "++" indicates that it is not "just" a bass drum voice. As is customary in all of Thomas' percussive voice modules, a considerable degree of control is included so that one is not locked into a particular drum sound. This handy module can produce not only bass drum voices, but also produces very fine low and high tom, wood block, and a variety of electronic drum sounds as well. And get this: this module does all of this from only two ICs and barely a handful of parts! (sMs Note: No calibration is necessary after building this module)

Just because the circuit is simple, don't confuse that with it being simplistic - this is a versatile full fledged drum voice in its own right, and sounds fantastic. Of the two designs that preceded it, the Mega Percussive Synthesizer (MPS) and the UD-1, the Bass++ is more closely related to the UD-1, though it does employ a circuit block trick used in the MPS.

Unlike the UD-1, which uses the lamentably disappearing NE566 VCO, the Bass++ uses parts all very common and obtainable in today's electronics scene. This is a project that can be built very easily on perfboard, and is not so complex that making more than one to fit into a single module is not anywhere close to being a Herculean task. As with all of Thomas' designs, it is very direct and nothing is wasted - in other words, it is an extremely elegant design.

The Bass++ Feature Set

Just writing this description continues to amaze me that Thomas pulled this whole thing off with just two ICs.

The Bass++ consists of a "Shell" VCO that establishes the pitch at which the simulated drum operates (the resonant filter of the synthesized drum cavity). Not only can that pitch be set manually, but it can be modulated over a variable range by a control voltage input. Moreover, this pitch can also be swept by the built in envelope generator, providing anything for the effect of the cavity/head flexing with each simulated drum beat to electronic drum "pyeeewwww" sound that is so ubiquitous in early '70s vintage disco music.

The envelope generator is triggered by a short pulse input (generally available in most analog synthesis equipment). A decay control allows the user to set the amount of time the drum voice lasts when it is "struck" by the input trigger.

The envelope generator also provides the impact portion of the synthesized drum hit, mimicking the effect of an object striking the synthesized drum voice. A tone control is provided for this signal so that the effect can be varied from a dull thump to a sharp crack.

The shell volume and impact volume can be mixed together in the proportion desired to fine tune the timbre of the synthesized percussive voice.

And, finally, a master volume control can be used to adjust the levels from 10Vp-p modular signals down to line level signals suitable for direct input into an amplifier.

A Bass++ Drum Voice Sound Sample

A sample of the Bass Plus-Plus triggered and pitch controlled by the Klee Sequencer can be found here (5.5 MB)

http://electro-music.com/forum/download.php?id=13003

This sample uses but one fixed set of controls, and demonstrates just one aspect of the sound of the Bass++. Hopefully it illustrates the fact that, though this circuit is a two IC design, the quality of the sound it produces is certainly up to the standards of any of Thomas' drum voices.

The Klee Sequencer is triggering the Bass++, while at the same time it is also controlling the shell pitch with a sequenced voltage output. The sequence range is shifted from low to high at various points so that the listener can hear the effect on the Bass++.

Bass++ Controls, Connectors and Indicators

Trigger Input/Envelope Generator Section

Trigger Input Jack: Accepts a trigger signal from external synthesizer equipment.

Sensitivity Control: Allows adjustment of the trigger input reference level so that the Bass++ responds consistently to the trigger level.

Status LED: Briefly pulses on with each trigger input for visual feedback of operation.

Decay Control: Allows adjustment of the length of time the drum voice envelope will take to fall away after each trigger input.

Shell VCO Control Section

Pitch Control: Sets the initial pitch of the Shell VCO.

Sweep Control: Sets how much effect the envelope generator will have on the pitch of the Shell VCO - in other words, how much, if any, of envelope sweep is allowed.

CV Jack: Accepts control voltages used to modulate the pitch of the Shell VCO.

Range Control: Sets the level of effect the applied control voltage will have on the pitch of the VCO - from none to full.

Impact Section

Impact Tone Control: Sets tone of the impact signal from bass to treble.

Signal Mix/Output Section

Shell Volume: Adjusts the amount of Shell signal present in the final output signal.
Impact Volume: Adjusts the amount of Impact signal present in the final output signal.

Master Volume: Adjusts the output signal level.
Output Jack: Provides the output signal.

Bass++ Circuit Theory and Operation

Here's a description of the Bass++ theory of operation in Thomas' own words. Please reference the circuit diagram in Figure 1 at the end of this description for clarity.

I've been designing electronic bass drums for some two dozen years now, but the Bass++ is by far the best, if I may be so immodest. It uses no hard-to-find parts, is very stable and predictable and is a snap to build, weighing in at two chips only. And since it employs legitimate analog synthesizer technology and not the old ringing filter routine, it is very quiet. Before continuing, let me hasten to add that the Bass++ is great for bass drums (of course), but also is perfect for tom-toms, woodblocks and related instruments thanks to its wide range. Let's dig in and see how it works. Refer to the schematic.

A trigger is applied to jack J1. It is assumed that this follows the usual Electronotes standard of being 0 to +5V in amplitude and 5 milliseconds wide. But if not, there is enough play in the "Sensitivity" control, R13, to accommodate most any signal. The output of IC2a will be a conditioned pulse, varying from 50 microseconds wide on up to 2.5 milliseconds, depending on the "Sensitivity" setting.

This signal is routed via D2 to the pulse stretcher composed of C8, R32 and Q2. With a greater duty cycle, LED D6 lights long enough to be readily seen and serves as a status indicator.

The VCO is made up of IC1a and associated components. This simple yet clever configuration comes straight from the LM13700 data sheet. C2 is the integrator timing cap. This can be changed to alter the audio range if desired. D4 and D5, along with R6 and R19 set the comparator trip points. The output is a triangle wave which is buffered internally by the circuitry connected to pins 7 and 8 of IC1a, with R9 biasing the buffer. Since this is a simple circuit, there is a slight pip at the extremes of the triangle, but this is hardly objectionable in a percussion circuit.

Strictly speaking, this is a current controlled oscillator. The current input is at pin 1 of IC1a. To make this simpler to use in a synthesizer setting, Q4 converts an input voltage to a current. The current will swing from 0 to 500 microamps, corresponding to an audio range of 10Hz to 5200Hz, perfect for drum work. And, oh, the 500 microamp max is on purpose---that's a good safe value to keep the OTA under so that it doesn't go into thermal runaway.

Since the internal buffers of the LM13700 introduce a negative offset, the output of the VCO is AC coupled by C14. The signal at this point is 10V peak-to-peak, so R24 and R2 drop this to 20mV peak-to-peak, which keeps the OTA operating in its linear region. R3 is provided to balance the differential input. Obviously, we're talking a VCA here, and the output current is converted to a voltage across R22. Observe C7. Its purpose is to smooth out the pip alluded to earlier. If you'd like a rougher edge to the sound, feel free to leave it out. The output of the VCA is then buffered by the internal circuitry at pins 10 and 9 of IC1b. Again, a capacitor (C15) blocks the negative bias, which also keeps the "Shell Volume" pot from sounding scratchy as you turn it; DC running through a pot is the equivalent of scraping your fingernails on a chalkboard. The signal finally winds its way to the mixer made up of IC2b.

Let's back up to the envelope generator (EG). The conditioned input trigger is fed to the EG by way of D1. C11 is the timing capacitor. A charge is rapidly dumped on to it for the attack, and then bleeds away through R4 and the "Decay" control R33. The envelope voltage is buffered by Q1. Notice that the entire voltage is developed across potentiometer R15. This voltage feeds R18 and is converted to a current by Q3. It is this current that controls the VCA mentioned above. As with

the other half of the chip, the maximum value of the control current is kept at about 500 microamps. Say what you will about my designs, at least they don't catch fire...

But the wiper of the "Sweep" control taps off a variable amount of the total voltage, and this is converted to a current by R23 and Q4. The current controls the VCO frequency as previously described. The emitter of Q4 is sitting almost at ground (close enough for state work), and we take advantage of this to form a quick-and-dirty mixer. That is, the "Sweep" control provides the moving signal, while the "Pitch" control, R29, blends in a constant offset. And for external control, J2 can be used to pump in a keyboard voltage, say, to make the drum tunable by hand. Again, the three controls (sweep, pitch and external) are mixed by R23, R20 and R21 respectively since the emitter of Q4 is sitting at "almost ground." It's an inexpensive but more than adequate technique for a drum.

And talking about cost effective, how about that impact generator? I learned this clever technique first hand from Craig Anderton, the guy who gave me my break in electronic music design and writing. (He used it to simplify one of my designs for _Polyphony_ magazine). The conditioned input pulse is coupled via D3 to a tone control network made up of C9, C10, R7 and R11. The pulse can be kept sharp or smoothed out for a bassier response by means of R14, the "Impact Tone" control. This is then applied to R31 which serves as the "Impact Volume" control. Note that R31 is ten times greater in value than R14; this is to prevent loading. I suppose a purist might turn his nose up at this approach, but I rather like it. It probably doesn't have quite the range the active impact generator in the Mega Percussive Synthesizer does, but then again it only uses a fraction of the parts. I've recorded some good music with this method, and in my book that's all that counts.

The impact sound is mixed with the shell (R25 and R26, respectively) via op-amp IC2b. The output is AC coupled by C13, and attenuated by the "Master Volume" control, R16. Remember, this is a synth level signal at 10V peak-to-peak, so tame it a bit if you're going straight to a line level device. Alternatively, you could always drop R27 to 20K if you'd like to permanently set the Bass++ for line level work.

The Bass++ is indeed a simple circuit, but I hope you'll come to appreciate some of the touches that make it a slick design (if I do say so myself). Parodying Einstein, I've tried to make things as simple as possible, but no simpler. In particular, the use of the "almost ground" node for mixing the control voltages, the virtually skeletal impact tone generator, the most uncomplicated VCO imaginable, the use of a mere two garden variety chips, along with the usual parts-reduction schemes that have always motivated me, all add up to a unit that is reliable yet a snap to build. Thanks to the low cost and ease of construction, four of these could be easily built without breaking the bank to handle a bass drum, a low and high tom-tom and woodblock. And don't forget metronomes! The Bass++ would make an admirable sound generator for such a circuit.

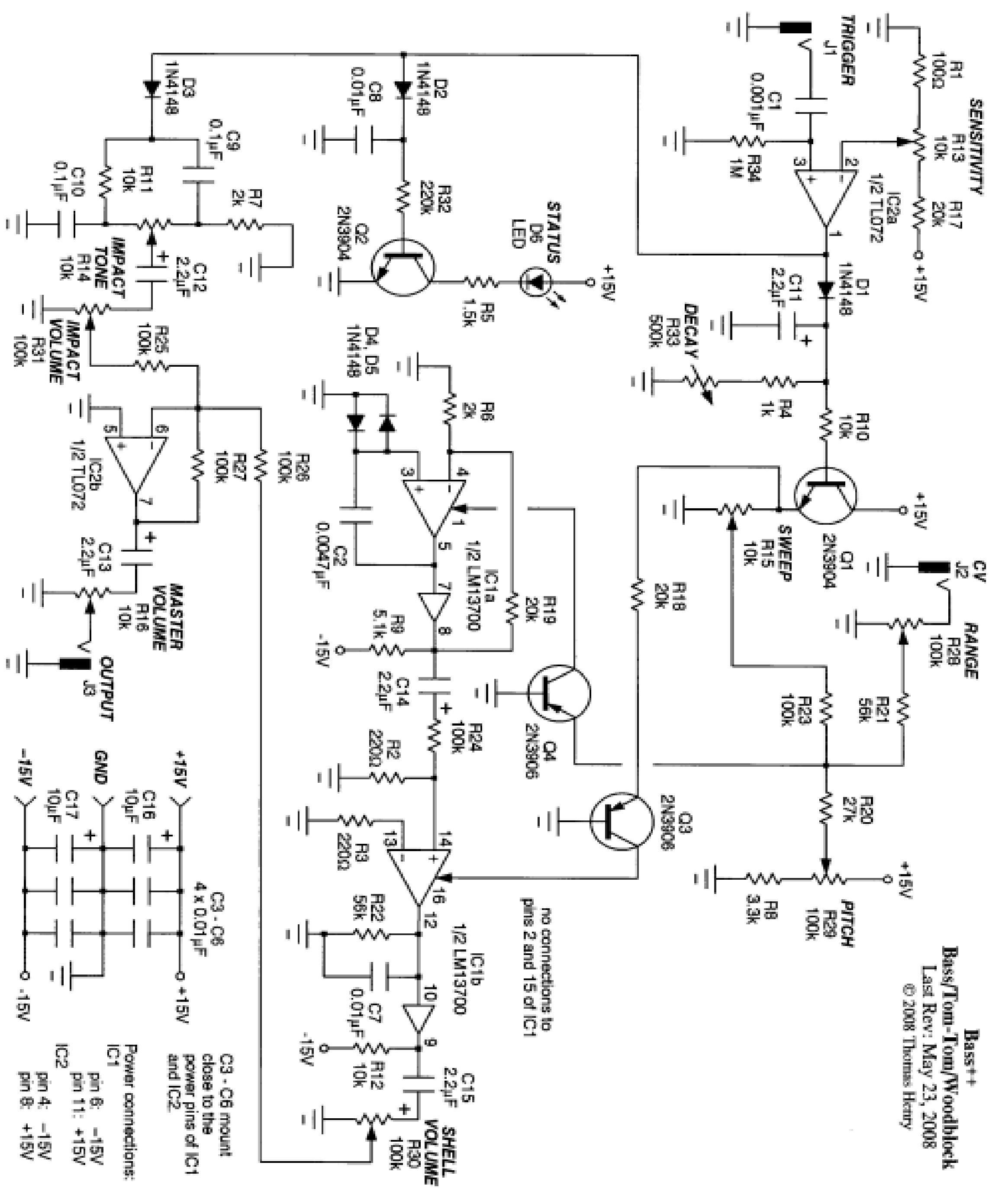


Figure 1 Bass ++ Schematic Diagram

The Bass++ Parts List

The parts list is shown in figure 2. The Bass++ parts are all very common and easy to find. A few good sources for a one stop search for ICs would be the following electronics web stores.

http://www.jameco.com/

http://www.digikey.com/

http://safe.dhwd.com/cgi-bin/debco/index.html

http://futurlec.com/index.shtml

***ALTERNATELY*: If you want to truly do one stop shopping, then you can purchase the Quad Bass++ PCB and parts kits from sMs Audio Electronics in single module increments (Module A,B,C,D) that not only include the parts listed in figure 2, but also includes IC sockets, ALL connector hardware with crimp terminals, panel control potentiometers, 1/4" or 1/8" panel jacks (please specify), and optional knobs (small additional fee).**

All fixed resistors are 1/4-watt, 5% values.

R1	100Ω
R2, R3	220Ω
R4	1k
R5	1.5k
R6, R7	2k
R8	3.3k
R9	5.1k
R10 – R12	10k
R13 – R15	10k linear potentiometer
R16	10k audio potentiometer
R17 – R19	20k
R20	27k
R21, R22	56k
R23 – R27	100k
R28, R29	100k linear potentiometer
R30, R31	100k audio potentiometer
R32	220k
R33	500k linear potentiometer
R34	1M

Miscellaneous: printed circuit board, IC sockets, front panel, knobs, wire, etc.

All capacitors are 16V unless noted.

C1	0.001μF mylar
C2	0.0047μF mylar
C3 – C6	0.01μF disc
C7	0.01μF mylar
C8 – C10	0.1μF mylar
C11 – C15	2.2μF electrolytic
C16, C17	10μF electrolytic

Semiconductors

D1 – D5	1N4148 diode
D6	LED
Q1, Q2	2N3904 NPN transistor
Q3, Q4	2N3906 PNP transistor
IC1	LM13700 dual OTA
IC2	TL072 dual op-amp

Other components

J1 – J3	1/4" phone jack, n.o.

Figure 2 Bass ++ Parts List

For the miscellaneous items, you can check out Futurlec for the IC sockets, 0.1" connector hardware, and potentiometers, Here is a list of part numbers and part descriptions The quantities are for each module section:

POLHDR6 6 Pin .100" Straight Male Polarized Headers (Qty 3)

POLHDCON6 6 Pin .100" Polarized Header Connector (Qty 3)

PLHDPIN Crimp Pin for Header Connector (2 Pkgs. Of 10 – Leaves 2 spare crimp pins)

ICS16 16 Pin IC Socket (Qty 1)

ICS8 8 Pin IC Socket (Qty 1)

POT10K 10K Linear Taper Pot (Qty 3)

POT10KA 10K Logarithmic Taper Pot (Qty 1)

POT100K 100K Linear Taper Pot (Qty 2)

POT100KA 100K Logarithmic Taper Pot (Qty 2)

POT500K 500K Linear Taper Pot (Qty 1)

The Quad Bass++ Concept
(a few words by Bill "State Machine" Manganaro of sMs Audio Electronics)

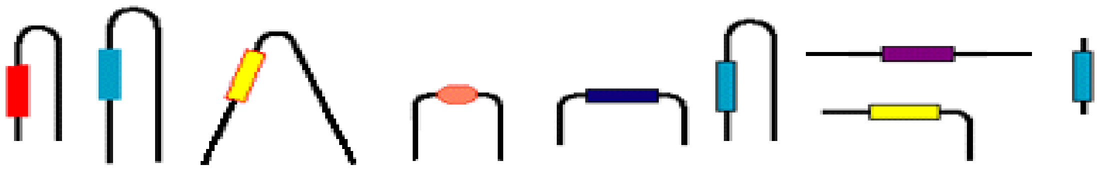
Worth mentioning once again, the Bass++ is a powerful drum voice with several front panel controls and jacks to control the various drum parameters. These controls can give the Bass++ a different sound at any given moment from deep bass to tom-tom to wood block sounds. After Dan "Antman" Lavin approached me about distributing the Bass++ and exchanging alternate ideas about PCB configurations, he and I settled on, and thought, it would be a rather cool idea to have more than one complete Bass++ drum module available on a single circuit board. Keeping each Bass++ independent of each other was an important goal, sharing only the circuit and panel power connections among them. This would create a fairly complete electronic drum "set". Having each drum voice independent allows the builder to built one, two, three, or four modules depending on their needs. The number of Bass++ drum modules we agreed to reside on a single circuit board would be four and thus the Quad Bass++ was born. Now that was a handy circuit we thought. Personally, when building drum modules, I had always built more than one and it only made sense that more than one should reside on a single circuit board. This saves space and the trouble of mounting several boards in your cabinet.

After we settled on the idea, we asked for, and received, the blessings from Thomas Henry and Scott Stites to go ahead with the Quad Bass++ project. Dan began the PC board layout process. I was to proof the artwork, and make any necessary changes. I would also create any special silkscreen markings on the PCB to aid in construction, and create the documentation. sMs Audio Electronics is now producing and distributing the Quad Bass++ PCB and parts kits. Each module or "voice" would need 9 front panel control potentiometers, 3 jacks, one panel mounted LED, parts for the PC board, and cable hardware. The board is small enough, approximately 4.75" square, to allow the user to build it into any standard enclosure.

The Quad Bass++ PC Board Build Phase

OK, so now you know what the Bass++ is, how it works, the controls and what they do. You have a pile of parts and your PC board sitting in front of you. Now you should be ready to start building your Quad Bas++ module. The first thing to build is your PC board and by now you know just how many Bass++ drums you're going to populate this thing with. I personally find stuffing and soldering printed circuit boards the most relaxing part of any build. Figure 3 assembly diagram on page 21 shows an illustration of the entire PC board revealing all the parts, and their reference designations, a road map of where all the parts go if you will. Note that there are 4 sections labeled A though D in the middle of the PC board. Each section is demarcated with a solid line enclosing each section forming 4 equal squares. These represent the four independent Bass++ modules. If you are building 2 modules, then you would logically populate section A and B. Remember the parts list is for one module so you will need to purchase enough parts for two. Looks more complicated then it ought to be when looking at the whole PC board right? Figure 4 shows only a single section. Now that's better! In this case, it's section A, the first section you will start building. If you choose to build up another section, all the instructions are the same but just use the corresponding figures for each section your building. The same check lists apply. I have created check lists that cover modules A though D in all tables for your convenience. All the part reference designations are the same for each module. One major warning though, please pay attention to the component orientations !!!! The figures for sections C and D do not show the power connectors J1 "a" and "b" and filter capacitors C16 & C17 as they will be installed last.

Construction



NOTE ABOUT ESD

It is generally good practice to wear ESD protection when working with electronic components to prevent possible damage or stress. Most electronics workbenches should have some form of ESD protection like wrist straps, cords, and an ESD mat as shown below. At the very least, make sure you have momentarily grounded yourself before handling the components of the board or the completed board assembly.

ESD strap, resistance cord, and mat

PCB Construction

The tools required for construction:

Soldering Iron with Chisel Tip No Bigger than Any Pad in PCB

Wire Stripper

Tin/Lead Solder

20 – 24 AWG Wire (Stranded)

RMA Flux

Diagonal Wire Cutters

Round Nose Pliers or Optional Component Lead Forming Tool

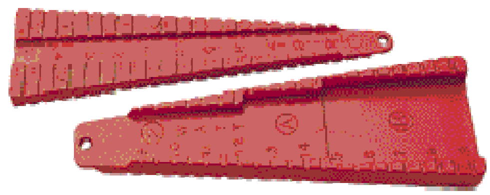
Optional IC Lead Forming Tool

Optional Flux Remover (Recommended)

Optional Magnifier (For Inspection)

RESISTOR INSTALLATION:

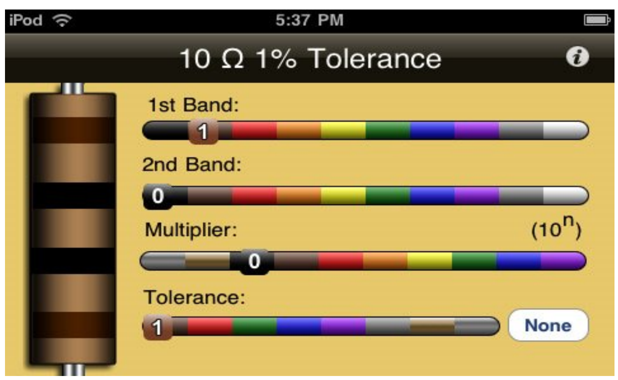
Start by sorting out and identifying all your components. Once you have identified your components, start by installing the fixed resistors. I find that using a lead bending tool works very nicely as shown below.



Lead forming "Christmas Tree"

You can also use round nose pliers to form the resistor leads. After forming the leads, install the resistors in accordance with the parts placement diagram (figures 4,5,6,7) applicable to the module your constructing and parts list provided in Figure 2.

Just in case you're rusty with resistor color codes, and you own an Apple "iPhone" or "iPod Touch", head over to Apples application store and get the "Resistor Color Code" application shown below for FREE ! Here is what the interface looks like …. Snazzy !! :



Place a resistor such that the body is flush against the PC board's surface. Once in place, bend the leads out at 45-degree angles on the solder side. Cut off the excess lead length with your diagonal cutters leaving about 1/16" of lead. With the resistors leads bent this way, they will not fall out and long leads will not be in the way when soldering. Do this for all the resistors then solder them all in place. Make sure your board has good topside solder flow as well as the bottom. Generally a bit of RMA flux works well to allow the solder to wick to the top of the printed circuit board. As each resistor is installed and soldered, check off the locations in Table 1 below.

MODULE A FIG 4	MODULE B FIG 5	MODULE C FIG 6	MODULE D FIG 7
R1 100R CHK ____	R1 100R CHK ____	R1 100R CHK ____	R1 100R CHK ____
R2 220R CHK ____	R2 220R CHK ____	R2 220R CHK ____	R2 220R CHK ____
R3 220R CHK ____	R3 220R CHK ____	R3 220R CHK ____	R3 220R CHK ____
R4 1K CHK ____	R4 1K CHK ____	R4 1K CHK ____	R4 1K CHK ____
R5 1.5K CHK ____	R5 1.5K CHK ____	R5 1.5K CHK ____	R5 1.5K CHK ____
R6 2K CHK ____	R6 2K CHK ____	R6 2K CHK ____	R6 2K CHK ____
R7 2K CHK ____	R7 2K CHK ____	R7 2K CHK ____	R7 2K CHK ____
R8 3.3K CHK ____	R8 3.3K CHK ____	R8 3.3K CHK ____	R8 3.3K CHK ____
R9 5.1K CHK ____	R9 5.1K CHK ____	R9 5.1K CHK ____	R9 5.1K CHK ____
R10 10K CHK ____	R10 10K CHK ____	R10 10K CHK ____	R10 10K CHK ____
R11 10K CHK ____	R11 10K CHK ____	R11 10K CHK ____	R11 10K CHK ____
R12 10K CHK ____	R12 10K CHK ____	R12 10K CHK ____	R12 10K CHK ____
R17 20K CHK ____	R17 20K CHK ____	R17 20K CHK ____	R17 20K CHK ____
R18 20K CHK ____	R18 20K CHK ____	R18 20K CHK ____	R18 20K CHK ____
R19 20K CHK ____	R19 20K CHK ____	R19 20K CHK ____	R19 20K CHK ____
R20 27K CHK ____	R20 27K CHK ____	R20 27K CHK ____	R20 27K CHK ____
R21 56K CHK ____	R21 56K CHK ____	R21 56K CHK ____	R21 56K CHK ____
R22 56K CHK ____	R22 56K CHK ____	R22 56K CHK ____	R22 56K CHK ____
R23 100K CHK ____	R23 100K CHK ____	R23 100K CHK ____	R23 100K CHK ____
R24 100K CHK ____	R24 100K CHK ____	R24 100K CHK ____	R24 100K CHK ____
R25 100K CHK ____	R25 100K CHK ____	R25 100K CHK ____	R25 100K CHK ____
R26 100K CHK ____	R26 100K CHK ____	R26 100K CHK ____	R26 100K CHK ____
R27 100K CHK ____	R27 100K CHK ____	R27 100K CHK ____	R27 100K CHK ____
R32 220K CHK ____	R32 220K CHK ____	R32 220K CHK ____	R32 220K CHK ____
R34 1M CHK ____	R34 1M CHK ____	R34 1M CHK ____	R34 1M CHK ____

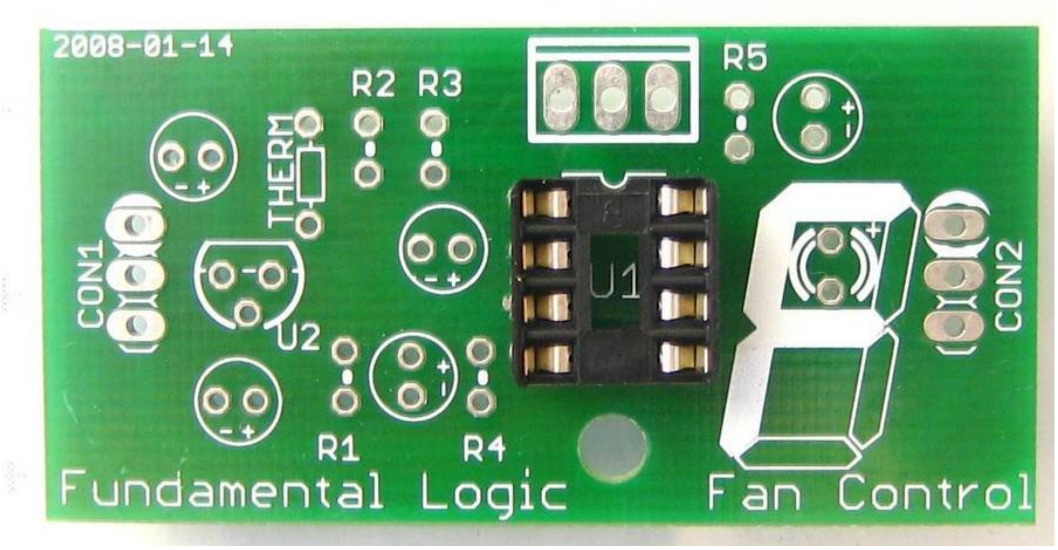
Table 1 Resistor Installation Checklist

DIODE – TRANSISTOR – SOCKET INSTALLATION:

Next install the diodes (D6 IS PANEL MOUNTED LED), transistors, and IC sockets (REFERENCED AS X1 & X2) according to Table 2. Be sure to observe the polarity of all these devices. The picture below illustrates where pin one is on the socket (the notched end). The notch end of the socket ends up on the same side where notch on the PC board silkscreen is.

The diode banded ends get placed into the square pad on the PC board.

Also, make sure the transistor flat side faces the flat side shown on the boards' silkscreen.



DIP IC Insertion Into PCB

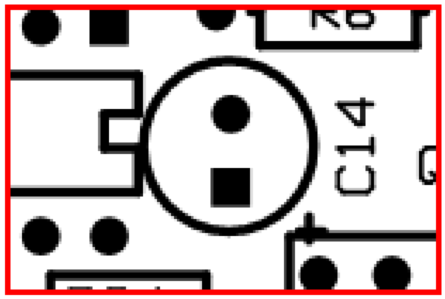
NOTE: Board shown is not the actual Quad Bass++ PC board and is just an <u>example</u> PC Board

	MODULE A FIG 4	MODULE B FIG 5	MODULE C FIG 6	MODULE D FIG 7
D1 1N4148	**CHK ____**	**CHK ____**	**CHK ____**	**CHK ____**
D2 1N4148	CHK ____	CHK ____	CHK ____	CHK ____
D3 1N4148	**CHK ____**	**CHK ____**	**CHK ____**	**CHK ____**
D4 1N4148	CHK ____	CHK ____	CHK ____	CHK ____
D5 1N4148	**CHK ____**	**CHK ____**	**CHK ____**	**CHK ____**
Q1 2N3904	CHK ____	CHK ____	CHK ____	CHK ____
Q2 2N3904	**CHK ____**	**CHK ____**	**CHK ____**	**CHK ____**
Q3 2N3906	CHK ____	CHK ____	CHK ____	CHK ____
Q4 2N3906	**CHK ____**	**CHK ____**	**CHK ____**	**CHK ____**
X1 16 PIN	CHK ____	CHK ____	CHK ____	CHK ____
X2 8 PIN	**CHK ____**	**CHK ____**	**CHK ____**	**CHK ____**

Table 2 Diode, Transistor, and Socket Installation Checklist

CAPACITOR INSTALLATION:

Next, install all the capacitors and headers according to Table 3 noting the polarity of the electrolytic capacitors and headers. These polarized capacitors are shown in red in the table. Note that the black striped side is the negative (-) terminal of the device as shown below. The positive side does not have this stripe. The headers are also shown in red. The locking tab for each one faces the inside of the PC board.



The positive lead of the electrolytic capacitors insert into the square pad on the PCB as shown in the above example. You can see C14 has a square pad. This is where the positive lead is inserted on the board. This is the same for ALL electrolytic capacitors. *<u>Note that C12 is not mounted to the board and is mounted on the front panel later in the build under the front panel wiring section.</u>*

The Mylar and ceramic capacitors are not polarized so no need to worry about their polarity when installing. This is typically how they will look.

Mylar Capacitors (No polarity)

Ceramic Capacitors (No polarity)

MODULE A FIG 4		MODULE B FIG 5		MODULE C FIG 6		MODULE D FIG 7	
C1 .001 Uf	**CHK ____**	**C1 0.001 uF**	**CHK ____**	**C1 0.001 uF**	**CHK ____**	**C1 0.001 uF**	**CHK ____**
C2.0047 uF	CHK ____	C2 0.0047 uF	CHK ____	C2 0.0047 uF	CHK ____	C2 0.0047 uF	CHK ____
C3 0.01 uF	**CHK ____**	**C3 0.01 uF**	**CHK ____**	**C3 0.01 uF**	**CHK ____**	**C3 0.01 uF**	**CHK ____**
C4 0.01 uF	CHK ____	C4 0.01 uF	CHK ____	C4 0.01 uF	CHK ____	C4 0.01 uF	CHK ____
C5 0.01 uF	**CHK ____**	**C5 0.01 uF**	**CHK ____**	**C5 0.01 uF**	**CHK ____**	**C5 0.01 uF**	**CHK ____**
C6 0.01 uF	CHK ____	C6 0.01 uF	CHK ____	C6 0.01 uF	CHK ____	C6 0.01 uF	CHK ____
C7 0.01 uF	**CHK ____**	**C7 0.01 uF**	**CHK ____**	**C7 0.01 uF**	**CHK ____**	**C7 0.01 uF**	**CHK ____**
C8 0.1 uF	CHK ____	C8 0.1 uF	CHK ____	C8 0.1 uF	CHK ____	C8 0.1 uF	CHK ____
C9 0.1 uF	**CHK ____**	**C9 0.1 uF**	**CHK ____**	**C9 0.1 uF**	**CHK ____**	**C9 0.1 uF**	**CHK ____**
C10 0.1 uF	CHK ____	C10 0.1 uF	CHK ____	C10 0.1 uF	CHK ____	C10 0.1 uF	CHK ____
C11 2.2 uF	**CHK ____**	**C11 2.2 uF**	**CHK ____**	**C11 2.2 uF**	**CHK ____**	**C11 2.2 uF**	**CHK ____**
C13 2.2 uF	CHK ____	C13 2.2 uF	CHK ____	C13 2.2 uF	CHK ____	C13 2.2 uF	CHK ____
C14 2.2 uF	**CHK ____**	**C14 2.2 uF**	**CHK ____**	**C14 2.2 uF**	**CHK ____**	**C14 2.2 uF**	**CHK ____**
C15 2.2 uF	CHK ____	C15 2.2 uF	CHK ____	C15 2.2 uF	CHK ____	C15 2.2 uF	CHK ____
J2a	**CHK ____**	**J2b**	**CHK ____**	**J2c**	**CHK ____**	**J2d**	**CHK ____**
J3a	CHK ____	J3b	CHK ____	J3c	CHK ____	J3d	CHK ____
J4a	**CHK ____**	**J4b**	**CHK ____**	**J4c**	**CHK ____**	**J4d**	**CHK ____**

Table 3 Capacitor & Header Installation Checklist

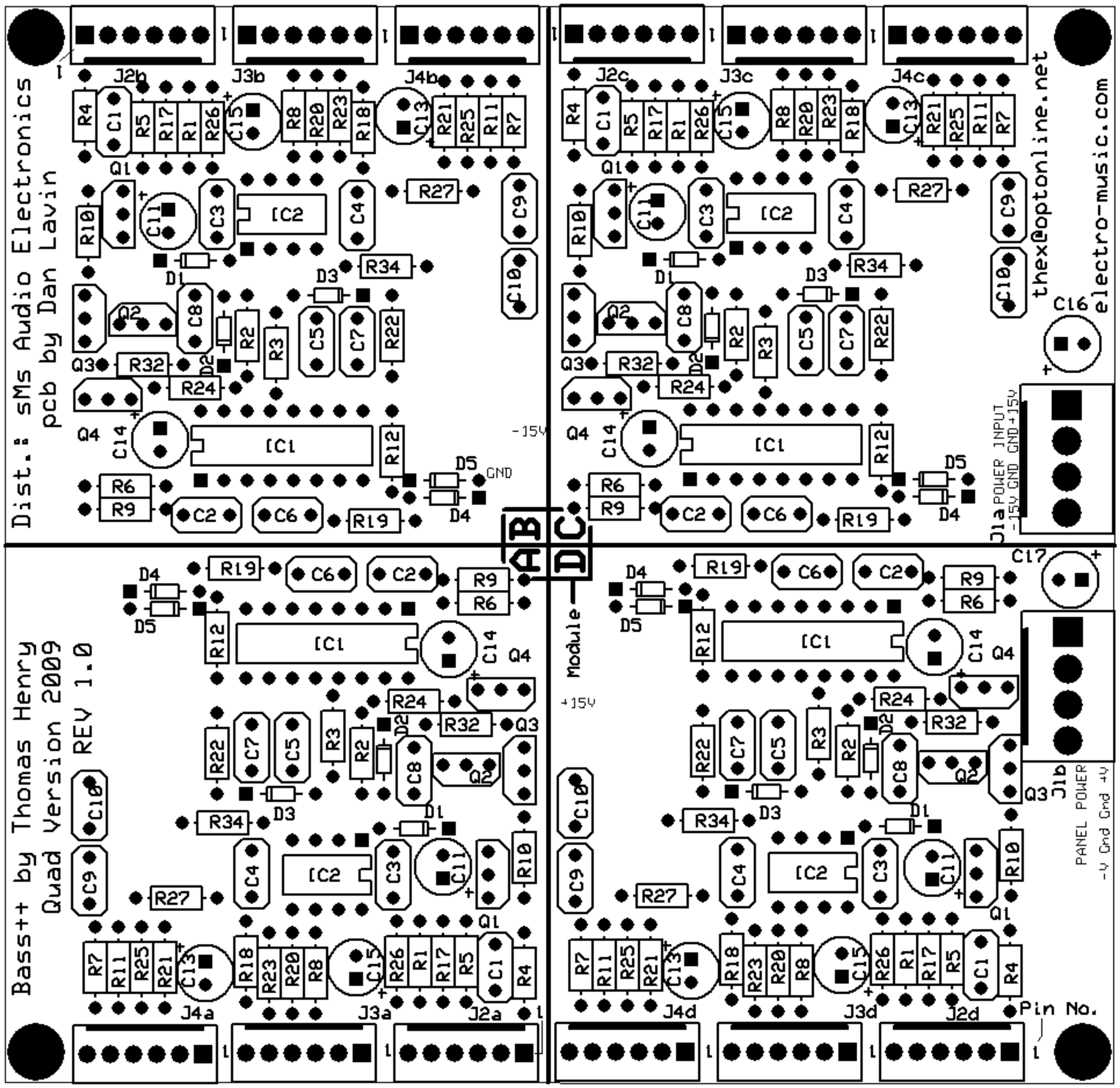


Figure 3 Assembly Diagram

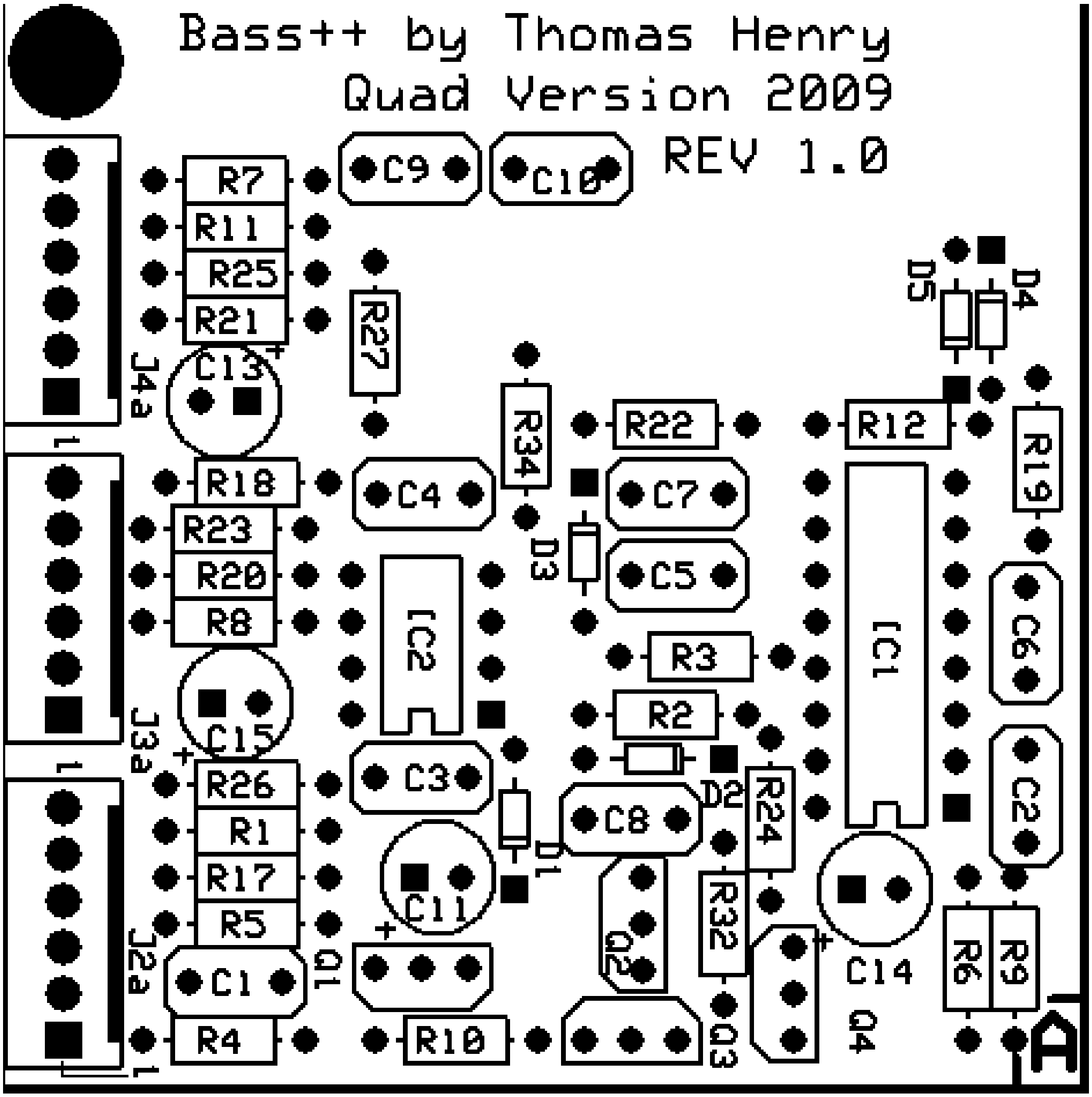


Figure 4 Single Bass++ Section "A"

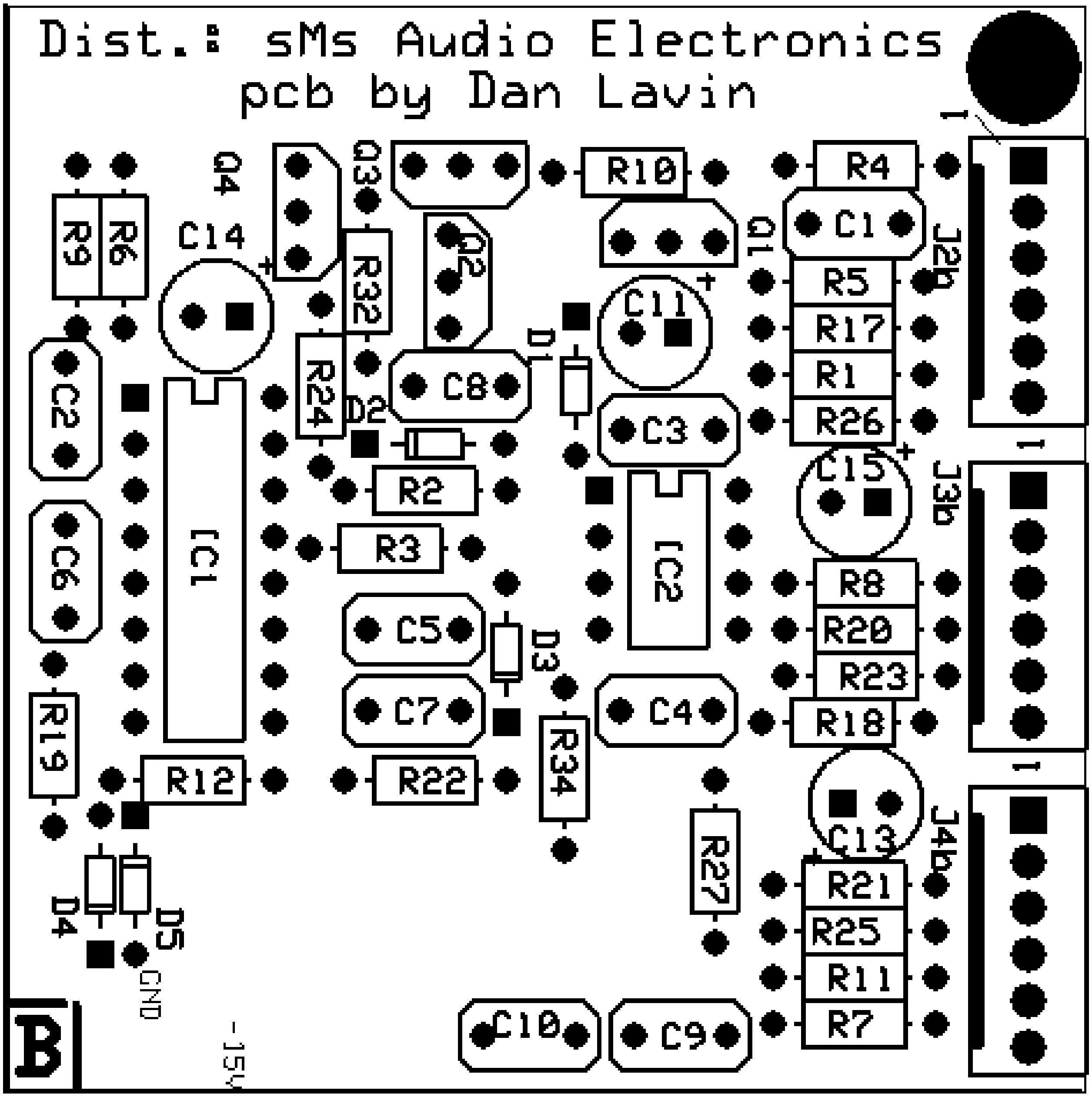


Figure 5 Single Bass++ Section “B”

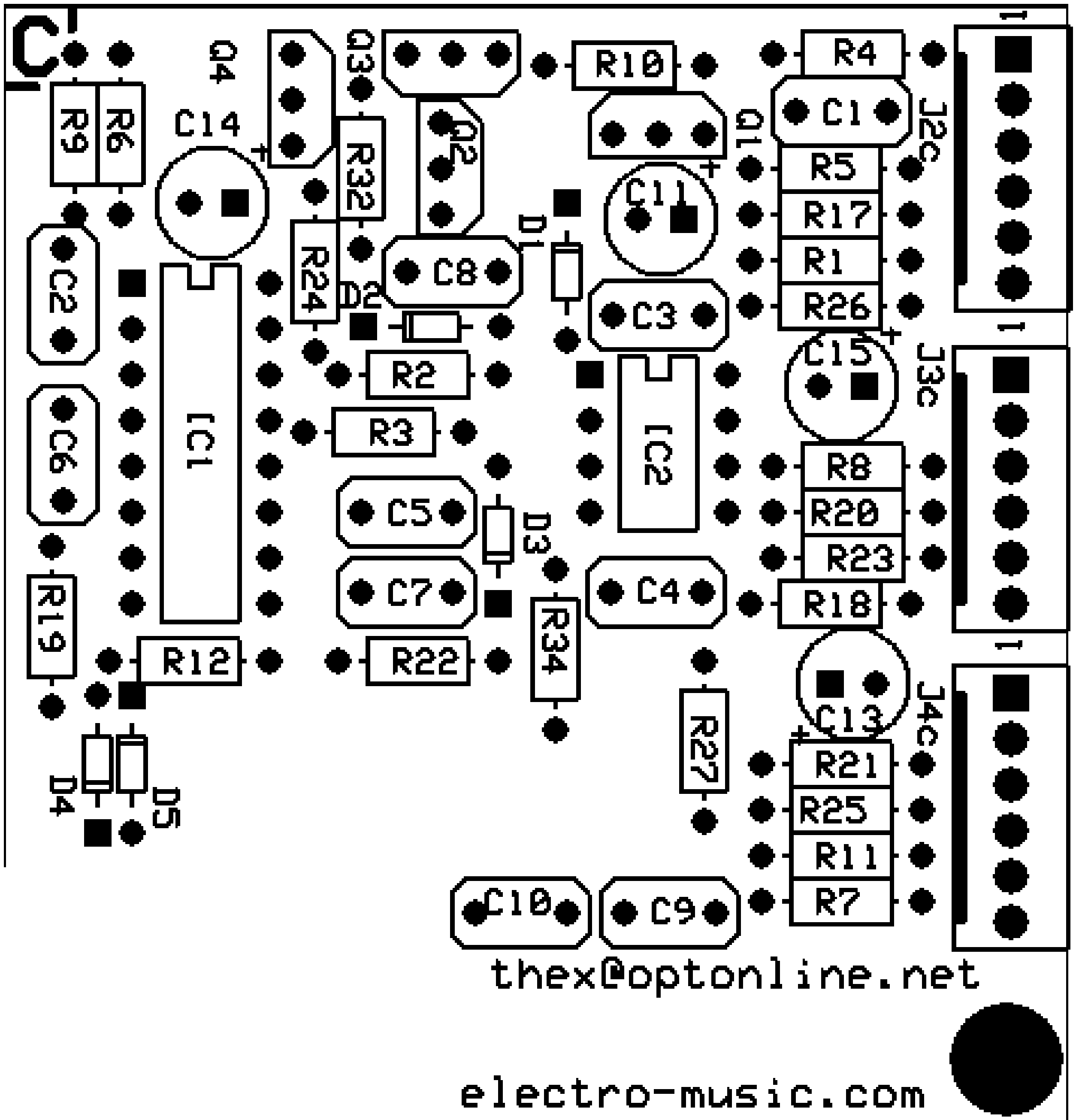


Figure 6 Single Bass++ Section "C"

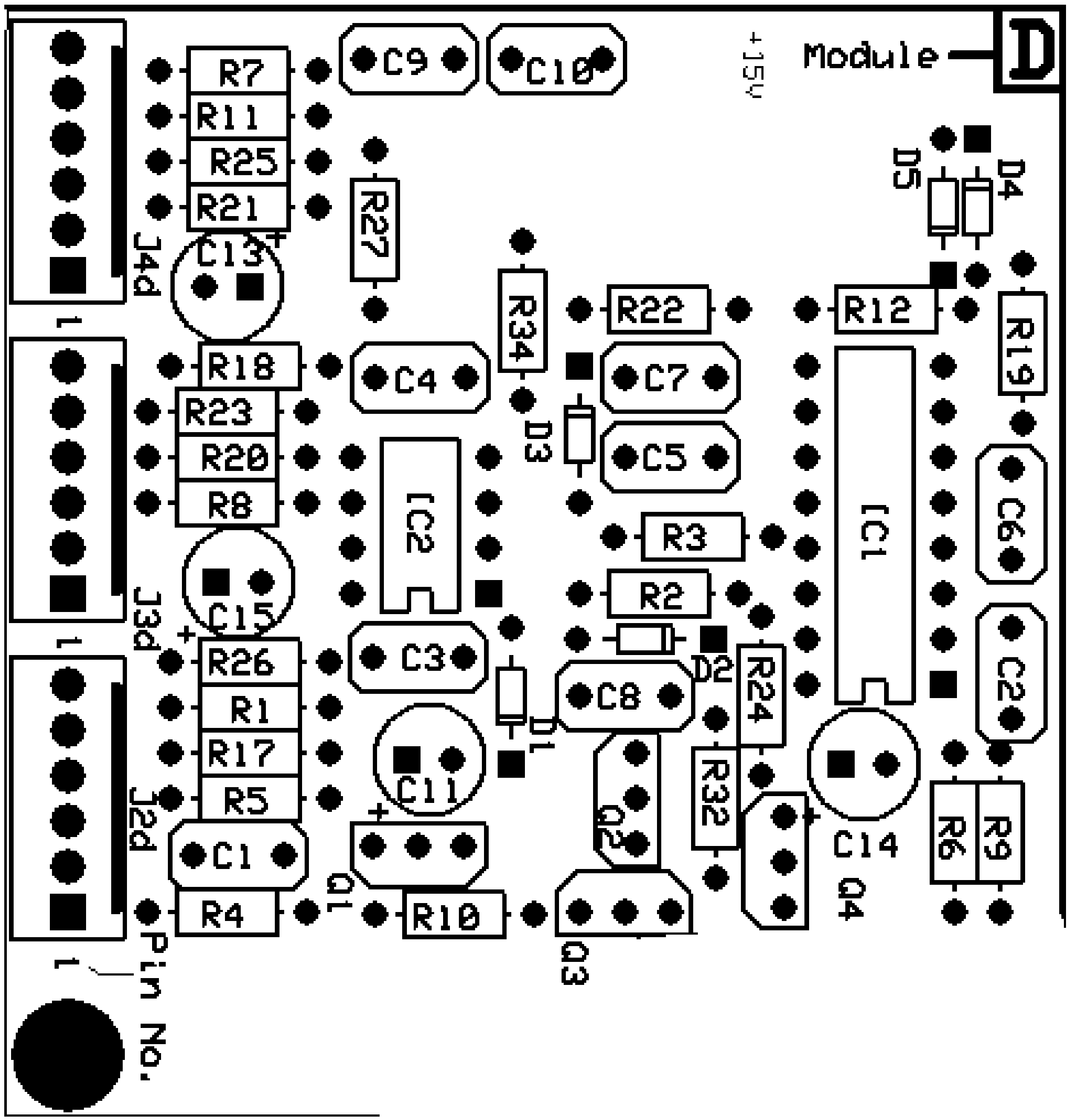


Figure 7 Single Bass++ Section "D"

POWER CONNECTOR AND FILTER CAPACITOR INSTALLATION:

This completes the drum module portion of the circuit and now it's time to install J1a, J1b, C16 and C17 according to Figure 8. Check off that you have installed these components in table 4. Make sure all the locking tabs are facing the inside of the board and that all electrolytic capacitor polarities are correct.

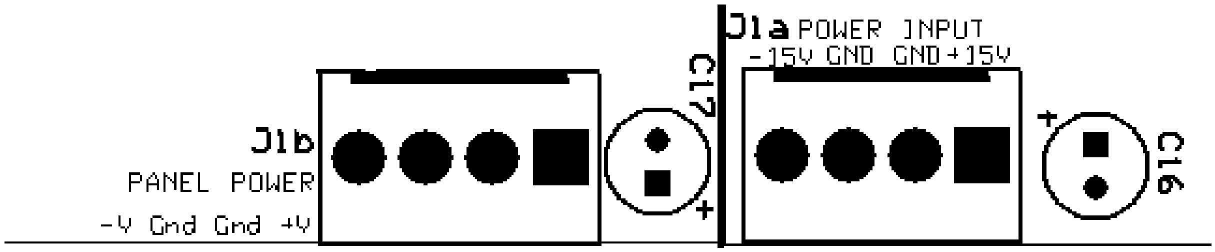


Figure 8 Single Bass++ Power Connectors and Filter Capacitors C16,17

J1a POWER INPUT	CHECK ______
J1b PANEL POWER	CHECK ______
C16 PSU FILTER CAPACITOR	CHECK ______
C17 PSU FILTER CAPACITOR	CHECK ______

Table 4 POWER CONNECTORS AMD PSU FILTER CAPACITORS CHECKLIST

CLEAN AND INSPECT BOARD:

After this, your board is complete, with the exception of the IC's being installed into the sockets. It is now a good time to check your work to make sure everything is in its right place and that ALL solder joints are good on the top and bottom of the board.

Next, be sure to clean all the flux off all the solder joints using flux remover or alcohol and use a brush on the difficult areas. Be sure to test a small amount of solvent on a small area of the board to be sure no plastic parts will melt or deform but I find all flux removers do not damage components. sMs uses "Pure-Tronics Flux Remover", manufactured by T.A. Emerald Industries shown below. It works well and does not damage any PCB components. It is best to clean the flux off the board as soon as you can. The longer you wait, the harder it is to remove the flux. Follow all the directions on the container closely and you will get very good results. Just PLEASE use in a very well ventilated area as the fumes can be harmful if your exposed to them too long. The flux remover can be found at "All Electronics" web store and at the time of this writing, a 13 OZ. can was $6.35 USD. The link for their web page is shown below. Just go to the chemicals section of their site to purchase this flux remover.

http://www.allelectronics.com/index.php

http://www.allelectronics.com/index.php

T.A. Emerald Industries Cleaning Products

IC INSERTION:

After the board is cleaned and inspected, it is time to install the IC's U1 (LM13700) and U2 (TL072). First, I find a lead straightening tool, like the one shown below, really comes in handy before inserting all DIP IC's. DIP IC pins are usually spaced wider than the spacing of the pins on sockets, which makes it very difficult to insert the IC's without bending leads. If you have this tool, place each IC in this lead straightener tool to put a nice right angle on all the pins.

Alternately, If you do not have a tool, place each side of the DIP IC against a flat, hard, surface and push down to give the pins more of a right angle. Once the board is cleaned, dry, and all the IC leads are formed, carefully install the IC's in their respective sockets. Be careful to observe all IC polarities. Now inspect the entire board one more time for soldering workmanship and accurate parts placement. Check that all electrolytic capacitor, transistor, diode, and header orientations are correct and make sure no solder bridges exist between pads. Using an ohmmeter, make sure that the 15V supplies are not shorted to ground. Make sure sufficient topside solder (component side) exists on all components where visible. When you are absolutely sure the board has been

constructed properly with ALL the correct parts in place and that no shorts exist, it is ready for application of power later on after your panel and cables are built. The following sections will now cover that.

Building the Cables for the Quad Bass++

The following sections will now cover these steps starting with the construction of you cable harnesses. Note that the illustrations show 8 position cables but you will be building 6 and 4 position cables.

When constructing your Quad Bass++, three main reasons why you should use cable components should be considered.

(1) Final Assembly of a Bass++ module is made much easier. If one is to hardwire the connections, the wires must be soldered to both the front panel and the PCB assembly. This involves point-to-point wiring which involves a considerable amount of effort over building the cables and wiring them to the front panel.

(2) Servicing/trouble-shooting the Bass++ is made easier – the panel can be easily disconnected and re-connected to and from the PCB.

(3) It's easier to organize the wiring in a methodical manner, which leads to a neater, no "rats nest" build.

This is not to say that building the cables cannot introduce problems – the cables must be built with care so they do not themselves create problems with intermittent connections.

Soldering the wires to the board and performing point to point wiring is really an option that should not be considered.

For each section of the Quad Bass++ module, there are three six position cable assemblies for signal input and outputs. Only two four position cable assemblies for circuit and panel power are required for all module instances.

Parts Used In the Cables

Each cable consists of three elements:

The housing, which is a shell that the cables are "plugged into and which connects to the proper connector on the PCBs. There two types of headers used in the Quad Bass++ module, they differ by the number of wires they will accommodate. They are six position headers with 0.1" spacing and four position headers with 0.156" spacing. The header pins are crimped or soldered or crimped and soldered to the ends of the wires, depending on your chosen technique. Generally 22 gauge or 24 gauge braided wire is required. Do not use solid core wire.

Housings and Pins

First of all, one must be aware that the housings will plug into the PCB headers in only one direction – this ensures the wire that is to attach to pin 1 of the PCB connector is the correct wire. So, you must become aware which "hole" in your housing will mate with pin 1. On the back of the

PCB, you will notice that each connector has a square pad to designate pin 1 of the connector. The PCB silkscreen also designates where pin 1 is also so please pay attention. The connector is keyed so that the housing can only be plugged in with one orientation. In other words, the header is keyed so that the housing, which has a mating "key" will plug into it. On the housing, the key is a ridge running along the edge of the "bottom of the housing".

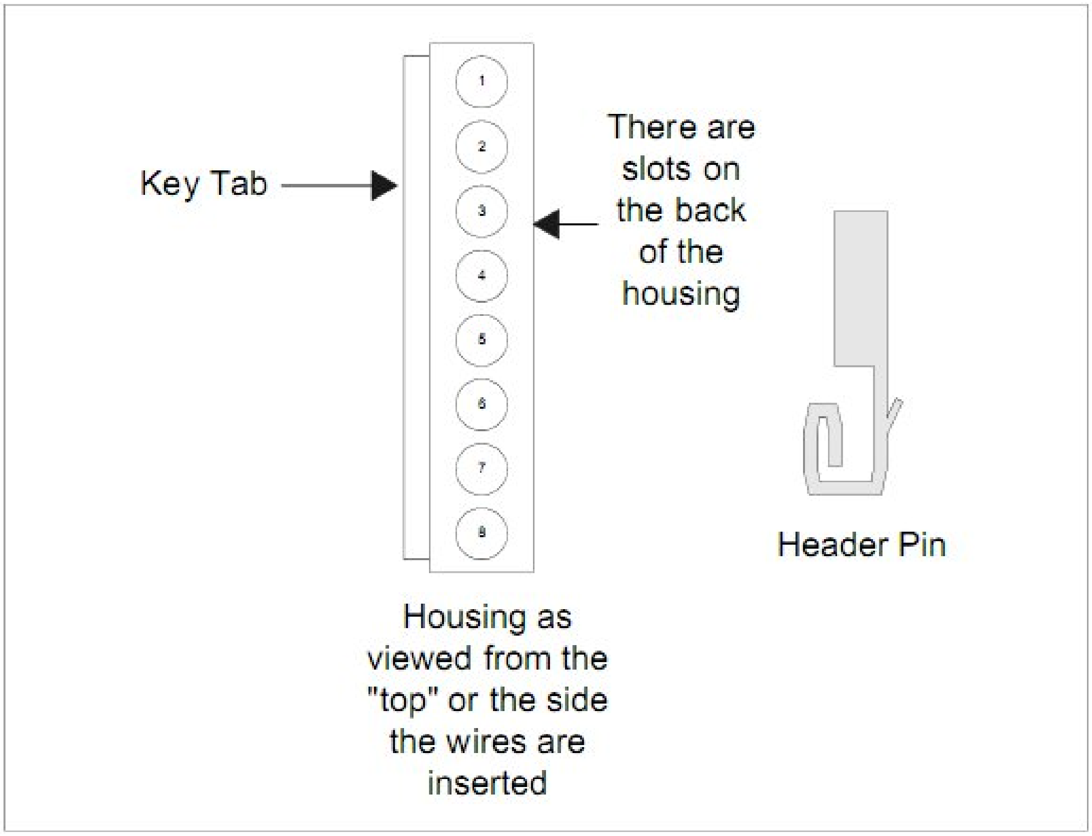


Figure 9 Housings and Header Pins

On the opposite side of the housing is a slot for each "hole" or terminal in the housing. These slots run along the bottom. The header pins, which attach to the ends of the wire, have a small tab that protrudes from the "back" of the pin. This small tab acts as sort of a fish hook and will "catch" in this slot when it is inserted into the housing.

Attaching the Pins to the Wires

The header pins attach to the ends of the wires, and are then inserted into the housing. The pins have a couple of tabs at the top that are intended to be crimped, or folded over the insulation of the wire. The end of the wire is stripped and tinned, about ¼" from the end of the wire. This

stripped and tinned part is to extend down so the “loop” of the header pin compresses down onto the bare wire when the wire and pin are inserted into the housing.

Crimping tools can be expensive, though fairly inexpensive types that resemble a pair of wire cutters are available. In lieu of a crimping tool, many people will fold the tabs over the bare wire and put a *small* amount of solder there to hold it in place (a large “blob” of solder may prevent the header pin from fitting into the housing). Generally, during the Quad Bass++ build process, it was found that the small amount of solder made a much more reliable connection.

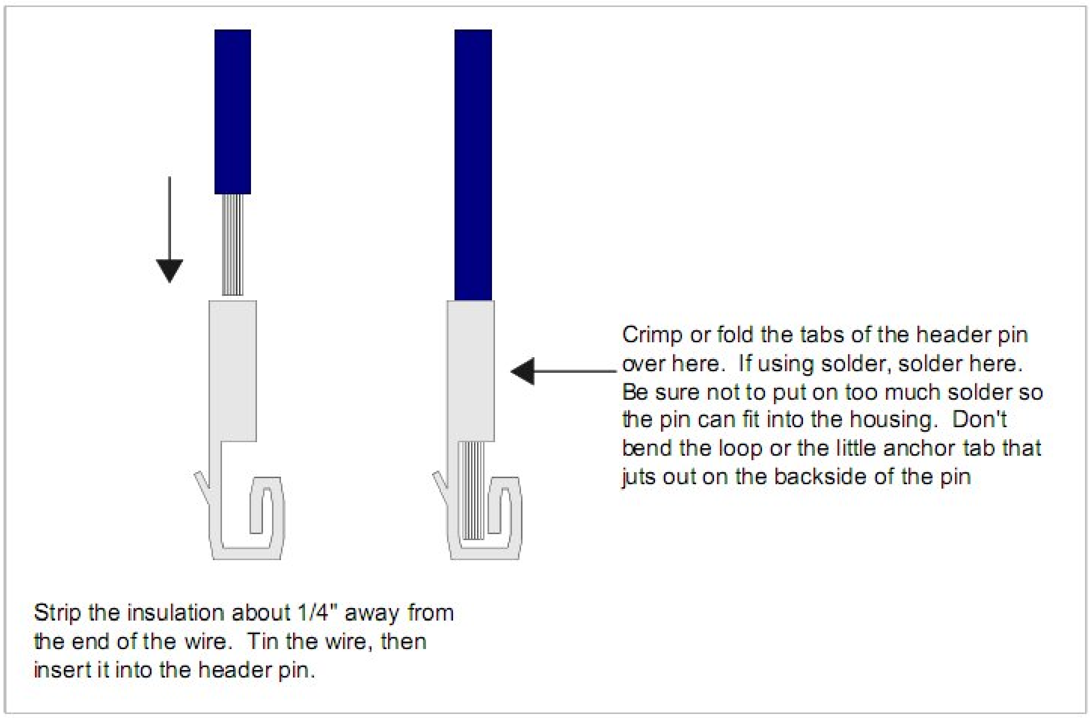


Figure 10 Putting The Header Pins On Wires

It’s a very good idea, if at all possible, to use a number of different colors of wire when building the cables. This helps you to determine very quickly which wire is pin 1, which wire is pin 2, etc. Otherwise, it’s very easy to become confused which wire is which, and this can lead to errors when soldering the free ends of the cables to the front panel.

Also, make sure you provide plenty of length for the wires. Make sure your wires are longer than they should be to accommodate any front panel you choose. By now, you’ve probably figured out how you’re going to mount the Quad Bass++ PCB in your chassis. You can break the job down into separate tasks. For example, you can go through task of stripping all of the wires needed and attaching the pins. Then you can task yourself with inserting the wire/terminal pin into the housings, and then you can task yourself with creating a neat looking cable by either wire-tying the cable wires together, or twisting the wires around each other to create a twisted-pair type thing. However, this last task may need to wait until after you’ve soldered the cables to the front panel – for now, it may be best to put a nice little twist-tie “collar” onto the cables just above the

housing once the cable is assembled. When you strip the wires, it's not a bad idea to strip just a tiny bit of the wire off the end that will connect to the panel. You can then use this exposed bit to attach a DMM or continuity checker to it and make sure your wire has continuity from the pin to the end of the wire. A little care at these early stages can save a lot of headache at the final assembly and test stage of building your Quad Bass++.

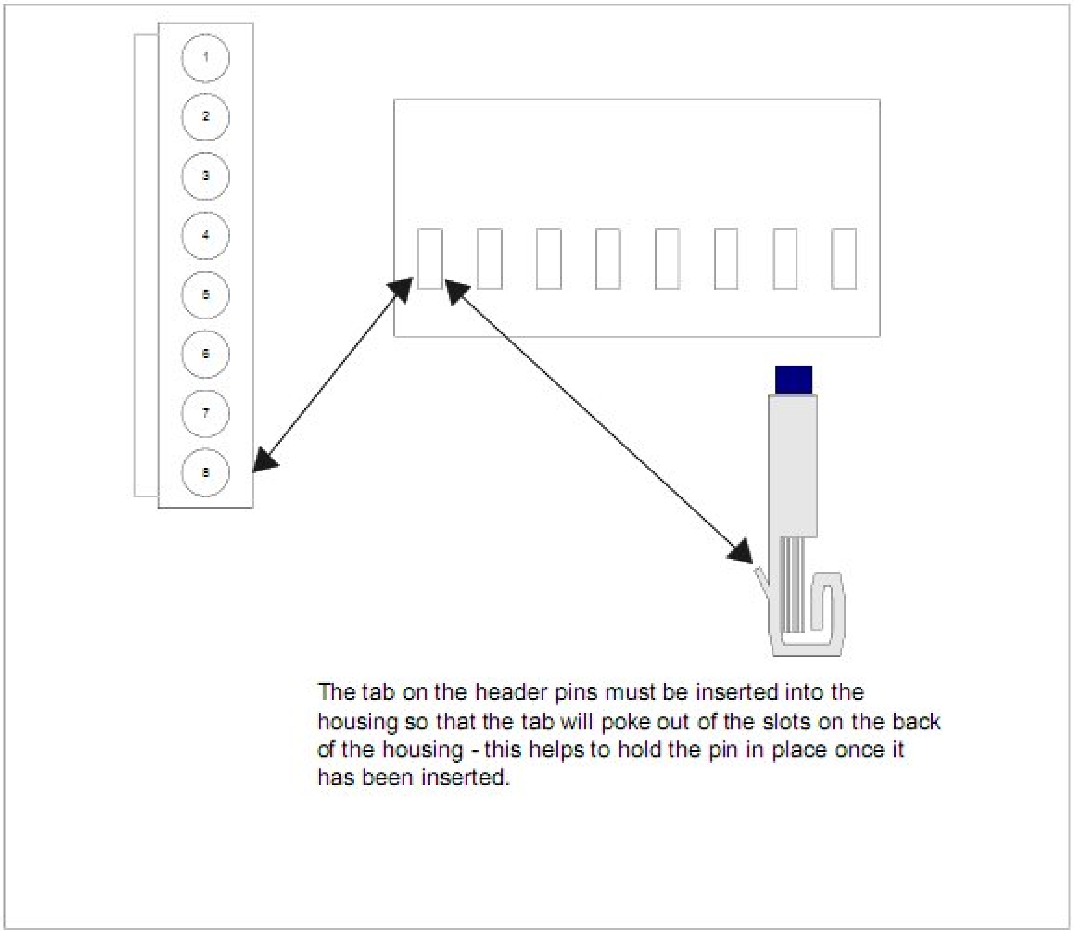


Figure 11 Pins Orientation Before Insertion Into the Housing

Once you've got the terminal pins attached to your wires, you can insert them into the housings. When you insert the pins into the housing, make sure that little barb on the back of the pin is oriented towards the side of the housing that has the small "window" in it – the barb is designed to "catch" in that window like a hook in the mouth of a fish, and make it harder to yank it out at a later date. When the pins are inserted into the housings, there should be the sensation of a soft "click" – this is a good sign that you've done a good job getting the pins on the ends of the wires.

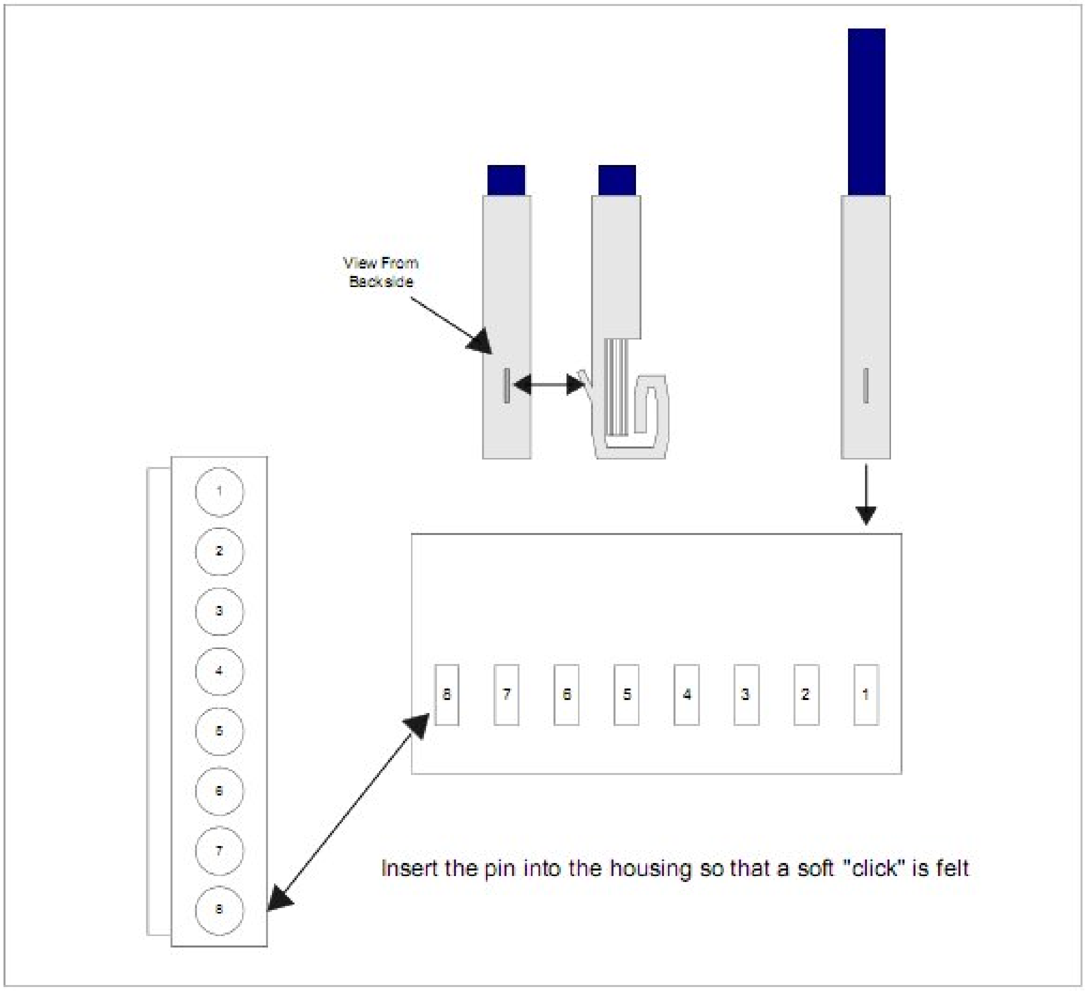


Figure 12 Inserting Pins Into the Housing

Remember that "loop" on the pin must not be crushed or misshapen – it is intended to compress down over the wire, making a good connection, and also provides a certain amount of tension that helps hold the pin in the housing. And, again, too much solder (if you put solder on) or solder in the wrong place can inhibit the actual installation of the pin into the housing.

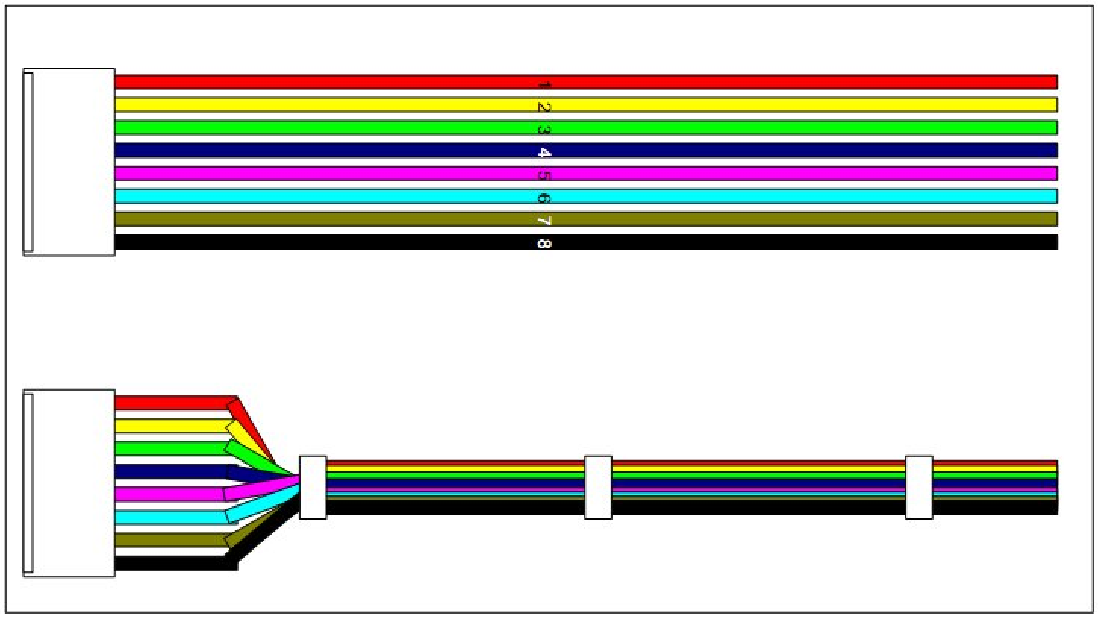


Figure 13 Assembled Cables

Once you've got all the wires into the housing, you can put a small collar onto the wires just above the housing. You may not want to add more at this time, because your wires will need to "fan out" in order to connect the all the disparate points they will connect to. Once you've built your cables, you will find that soldering them to your panel is a real cinch – very easy, and all the real work will be behind you.

Label the two 4 position power connectors P1a and P1b. Label, for example for module A, the other connectors P2a, P3a, and P4a. This can be as simple as using a fine permanent marker on the connector housing or using a label maker …. Your choice but it's a good idea to ID the ends.

The Front Panel and Wiring

By now you have your Quad Bass++ PC board fully assembled, you have inspected your work, and your cable assemblies are constructed.

The front panel upon which this project is mounted is a very personal thing and thus is left up to you to design. There are many form factors to choose from such as MOTM, Frac. Rack, 19" rack mounted, Eurorack, etc. You can even use your own proprietary form factor if you wish. While each panel can have different layouts such as colors, jack types, knob styles, layout, etc, the one thing that is constant is where the front panel components (potentiometers, jacks, LED) connect to the Quad Bass++ PC board's connectors. Table 4 below shows all the connections that are established between the front panel and the PC board. The connections in table 6 have been translated to Figure 14, an interconnect diagram, which shows the front panel wiring from the PC board to the potentiometers, jacks, and LED. This is one diagram from which you will work from and is easier to read than a list of wires. This diagram is further broken down into smaller wiring jobs to minimize error. Keep in mind, the diagram does not represent where you put your panel components, just how to connect them, no matter where you place them. IMPORTANT NOTE …..

Please observe that pin 1 of all the connector headers shown on the PC board. Please don't mistake pin 8 for pin 1 and end up wiring things backwards. I mention this because I have done this myself. Also, note the reference (middle right of drawing) in Figure 14 showing where the potentiometer TOP, WIPER, and BOT connections are located when looking at the potentiometer from the wiring side of the panel or back of the potentiometer.

Mount all your panel components to your front panel and then label the back of each potentiometer and with their reference designations using a sharpie marker or equivalent permanent marker. For the jacks, just mark the reference designation next to the part. This will avoid wiring mistakes. I do this personally to avoid error in wiring to the wrong panel potentiometer or jack.

Before wiring the cable to your panel, it's a good time to wire all the grounds together first. You can daisy chain, preferably a black wire (20 to 24 AWG). Use the following check list in Table 5 AND Figure 15 as you wire all your grounds together. Check off each wire placement as it's connected in the chain. For clarity, Figure 15 shows how the grounds should all be tied together. The ground connections are shown in GREEN for clarity.

R33 GROUND (BOT TERMINAL)	**CHECK ______**
R28 GROUND (BOT TERMINAL)	**CHECK ______**
R16 GROUND (BOT TERMINAL)	**CHECK ______**
R15 GROUND (BOT TERMINAL)	**CHECK ______**
R31 GROUND (BOT TERMINAL)	**CHECK ______**
R30 GROUND (BOT TERMINAL)	**CHECK ______**
J1 GROUND (SLEEVE TERMINAL)	**CHECK ______**
J2 GROUND (SLEEVE TERMINAL)	**CHECK ______**
J3 GROUND (SLEEVE TERMINAL)	

	CHECK ______
P1b- 2 & 3 TIE TOGETHER	CHECK ______
P1b-2,3 TO DAISY CHAINED GROUNDS (SHORTEST LENGTH)	CHECK ______

Table 5 Front Panel GROUND Wiring Checklist

Now cut the wires at P1b-2 and P1b-3 to about a 2” length. Tie them together and add one wire off this bundle that will be long enough to reach one of the above ground points that will create the shortest run to the panel. Solder the wires and insulate using tape of shrink tubing. Remember that if you use shrink tubing, the tubing goes on first before you solder the lead wire to the panel. Again, I mention this because I have made these very same mistakes.

CONNECTOR REFERENCE x = module [a] [b] [c] [d]	*SIGNAL NAME & CONNECTION*
J2x SIGNAL CONNECTOR	
J2x-1	DECAY POTENTIOMETER (R33) –TOP
J2x-2	TRIGGER J1 – TIP LUG
J2x-3	STATUS LED CATHODE (-)
J2x-4	SENSITIVITY POTENTIOMETER (R13) –TOP
J2x-5	SENSITIVITY POTENTIOMETER (R13) – WIPER
J2x-6	SENSITIVITY POTENTIOMETER (R13) –BOT
J3x SIGNAL CONNECTOR	
J3x-1	SHELL VOLUME POTENTIOMETER (R30) – WIPER
J3x-2	SHELL VOLUME POTENTIOMETER (R30) – TOP
J3x-3	PITCH POTENTIOMETER (R29) – BOT
J3x-4	PITCH POTENTIOMETER (R29) – WIPER
J3x-5	SWEEP POTENTIOMETER (R15) – WIPER
J3x-6	SWEEP POTENTIOMETER (R15) – TOP
J4x SIGNAL CONNECTOR	
J4x-1	MASTER VOLUME POTENTIOMETER (R16) – TOP
J4x-2	RANGE POTENTIOMETER (R28) – WIPER
J4x-3	IMPACT VOLUME POTENTIOMETER (R31) – WIPER
J4x-4	IMPACT TONE POTENTIOMETER (R14) – BOT
J4x-5	UNUSED
J4x-6	IMPACT TONE POTENTIOMETER (R14) – TOP
J1a POWER INPUT	
J1a-1	+15 VDC INPUT
J1a-2	GROUND
J1a-3	GROUND
J1a-4	-15 VDC INPUT
J1b PANEL POWER	
J1b-1	+15 VDC OUTPUT
J1b-2	GROUND
J1b-3	GROUND
J1b-4	-15 VDC OUTPUT

Table 6 Board Connector I/O Pin Assignments

Bass++ FRONT FULL PANEL WIRING DIAGRAM for (1) Module

NOTE: Modules A, B,C and D are all wired identically as shown

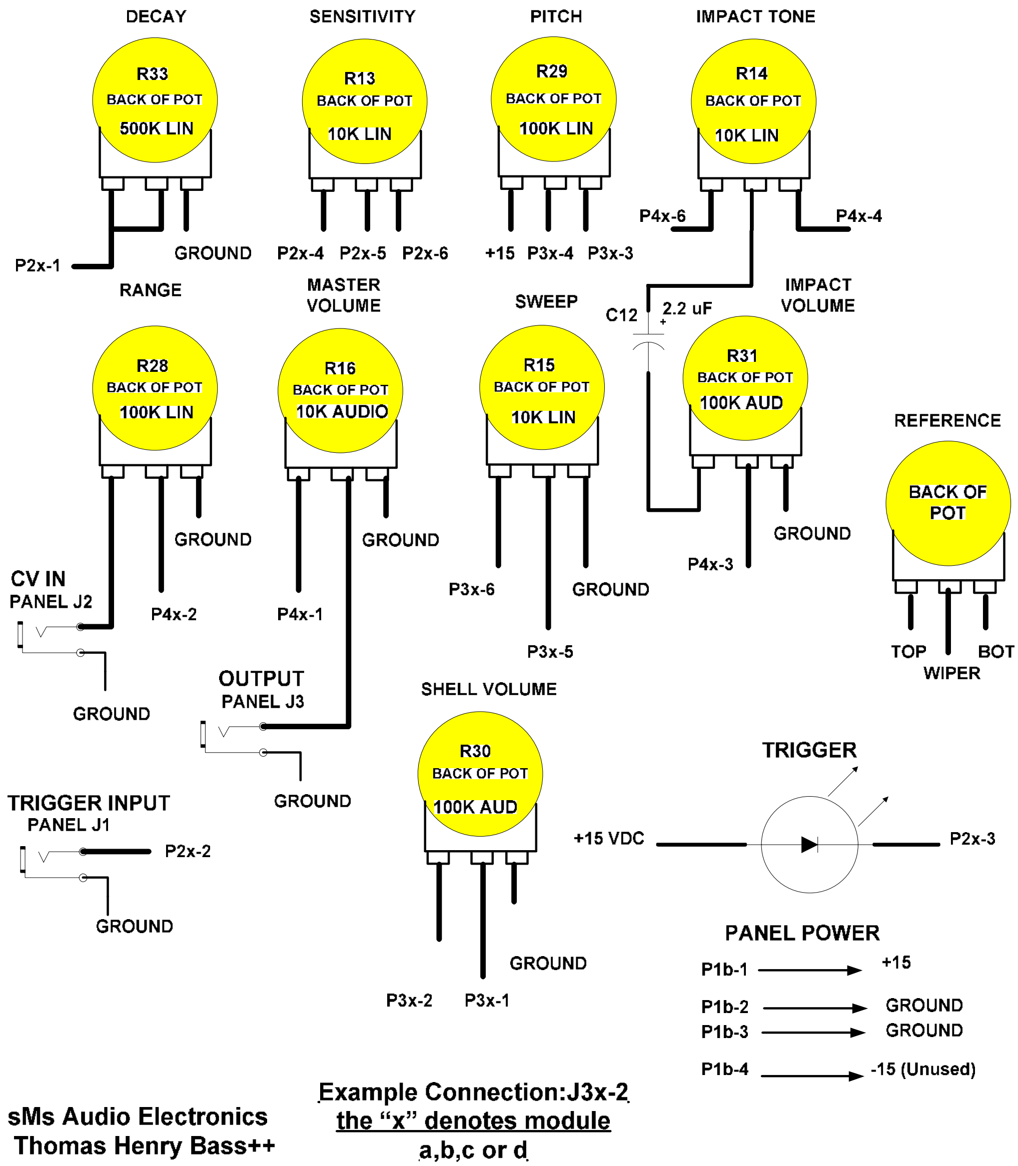


Figure 14

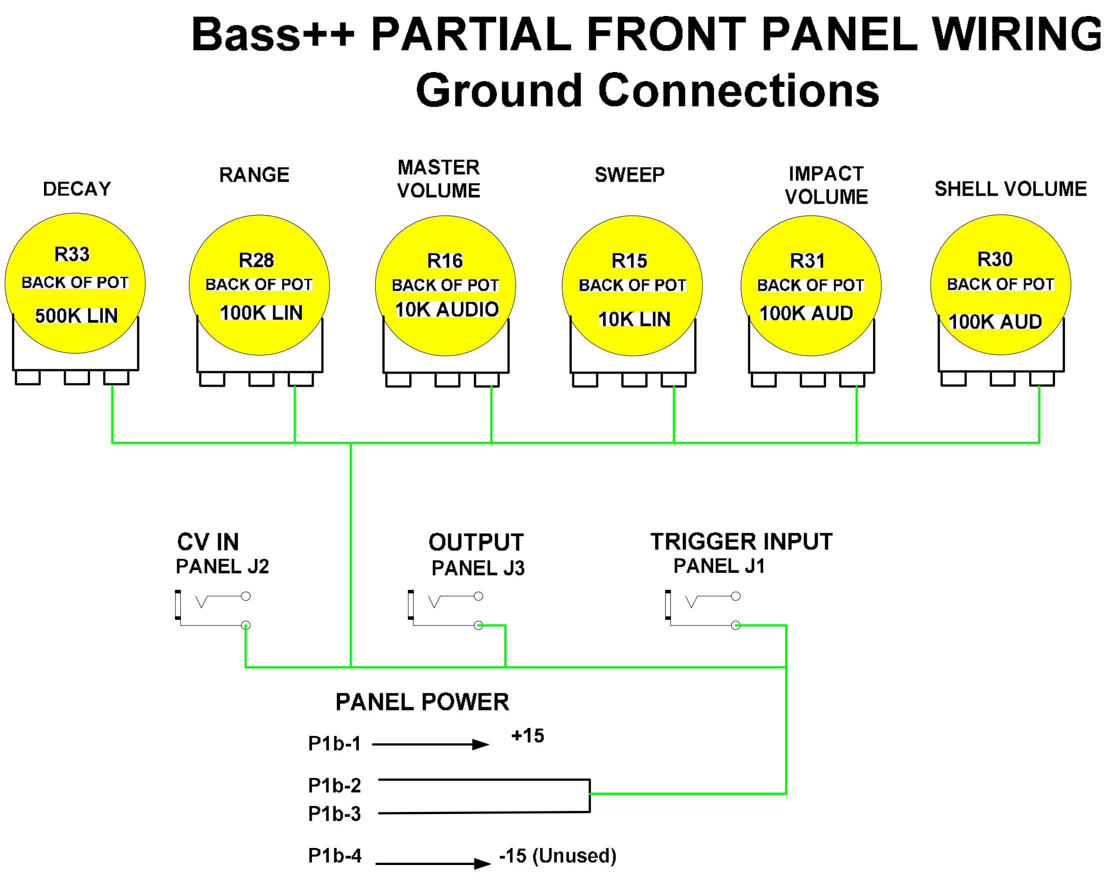


Figure 15 Ground Connections

Wire the +15V connections as shown in Table 7.

Run a wire from P1b-1 to one of the above +15V points that will create the shortest run to the panel.

For clarity, Figure 16 shows how the +15V is wired together. The +15V connections are shown in RED. Note the orientation of the LED. The rounded side (ANODE +) is the side that gets wired to the +15V supply line. The flat side (CATHODE -)will get wired later.

R29 +15V (TOP) to: LED +15V (ANODE TERMINAL)	CHECK ______
P1b-1 to LED ANODE TERMINAL OR R29 TOP (SHORTEST LENGTH)	CHECK ______

Table 7 (+15) Panel Wiring Checklist

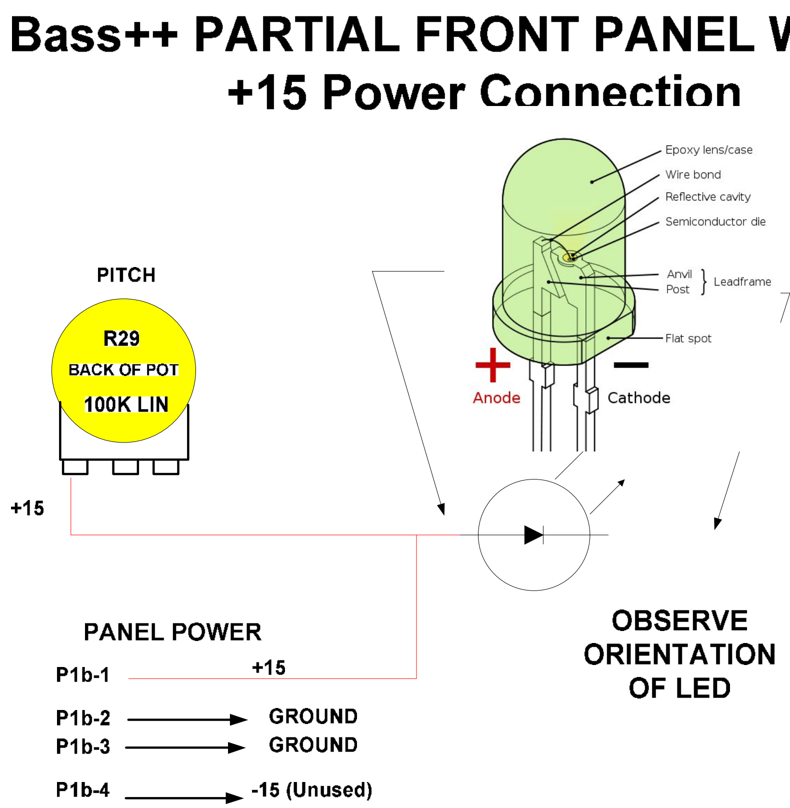


Figure 16 (+15) Connections

Next, install C12 on the front panel between R14 and R31 as shown in Figure 17. Please observe the part polarity. Use the checklist in Table 8 as you wire each end. These potentiometers should be next to each other as they both relate to the TONE controls and would eliminate the need to add wires to C12 when installing. Next add a jumper from R33-TOP to R33-WIPER. Check off in Table 8 when complete.

C12 + POSITIVE LEAD to R14 WIPER	CHECK ______
C12 – NEGATIVE LEAD to R31 TOP	CHECK ______
R33 TOP to WIPER JUMPER WIRE	CHECK ______

Table 8 (C12) & R33 Jumper Wiring Checklist

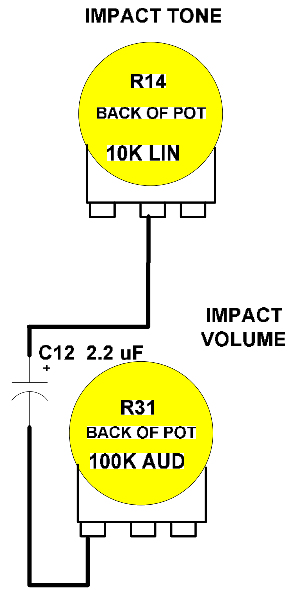
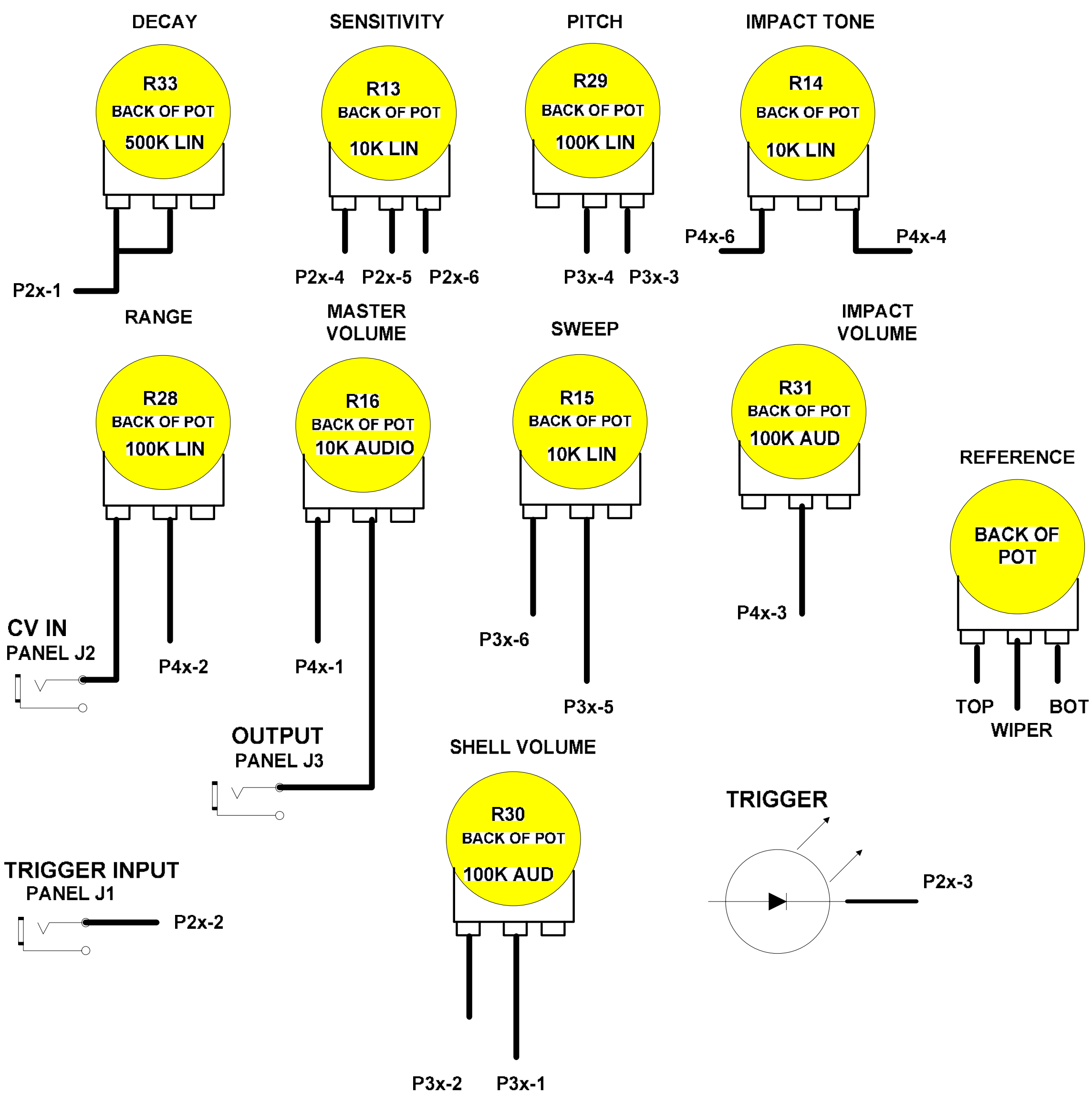


Figure 17 (C12) Connections

Now it's time to wire the P2, P3 and P4 connectors to the front panel. Figure 18 shows all the wires that need to be connected in RED. Use checklist Tables 9, 10, 11, or 12 depending on which module you are wiring (A,B,C OR D) and check off each wire as you place them to the panel. Now wire the last two jacks, J2 and J3. Use checklist Table 13 to check off each wire as you place them to the panel. Try to work neatly and cut the wires such that you don't have a "birds nest" of wires but enough service length to make any repairs in the future. If it's at all possible, mount the circuit board so that you can run the wire and cut it to its final length that it will be. If you can't mount the board, then lay it as close to its final position so that proper wire lengths can be determined.

Bass++ PARTIAL FRONT PANEL WIRING
P2x,P3x & P4x for (1) Module

NOTE: Modules A, B,C and D are all wired identically as shown



sMs Audio Electronics
Thomas Henry Bass++

Example Connection:J3x-2
the "x" denotes module
a,b,c or d

Figure 18 P2x,P3x,P4x TO PANEL WIRING

MODULE "A" CONNECTOR WIRING CHECKLIST		
CONNECTOR REFERENCE	*SIGNAL NAME & CONNECTION*	*CHECK IF INSTALLED*
P2a SIGNAL CONNECTOR		
P2a-1	**DECAY POTENTIOMETER (R33) – TOP**	**CHECK_____**
P2a-2	**TRIGGER J1 – TIP LUG**	**CHECK_____**
P2a-3	**STATUS LED CATHODE (-)**	**CHECK_____**
P2a-4	**SENSITIVITY POTENTIOMETER (R13) –TOP**	**CHECK_____**
P2a-5	**SENSITIVITY POTENTIOMETER (R13) – WIPER**	**CHECK_____**
P2a-6	**SENSITIVITY POTENTIOMETER (R13) –BOT**	**CHECK_____**
P3a SIGNAL CONNECTOR		
P3a-1	**SHELL VOLUME POTENTIOMETER (R30) – WIPER**	**CHECK_____**
P3a-2	**SHELL VOLUME POTENTIOMETER (R30) – TOP**	**CHECK_____**
P3a-3	**PITCH POTENTIOMETER (R29) – BOT**	**CHECK_____**
P3a-4	**PITCH POTENTIOMETER (R29) – WIPER**	**CHECK_____**
P3a-5	**SWEEP POTENTIOMETER (R15) – WIPER**	**CHECK_____**
P3a-6	**SWEEP POTENTIOMETER (R15) – TOP**	**CHECK_____**
P4a SIGNAL CONNECTOR		
P4a-1	**MASTER VOLUME POTENTIOMETER (R16) – TOP**	**CHECK_____**
P4a-2	**RANGE POTENTIOMETER (R28) – WIPER**	**CHECK_____**
P4a-3	**IMPACT VOLUME POTENTIOMETER (R31) – WIPER**	**CHECK_____**
P4a-4	**IMPACT TONE POTENTIOMETER (R14) – BOT**	**CHECK_____**
P4a-5	**UNUSED**	
P4a-6	**IMPACT TONE POTENTIOMETER (R14) – TOP**	**CHECK_____**

Table 9 Module "A" Panel Wiring Checklist

MODULE "B" CONNECTOR WIRING CHECKLIST		
CONNECTOR REFERENCE	SIGNAL NAME & CONNECTION	CHECK IF INSTALLED
P2b SIGNAL CONNECTOR		
P2b-1	DECAY POTENTIOMETER (R33) – TOP	CHECK_____
P2b-2	TRIGGER J1 – TIP LUG	CHECK_____
P2b-3	STATUS LED CATHODE (-)	CHECK_____
P2b-4	SENSITIVITY POTENTIOMETER (R13) –TOP	CHECK_____
P2b-5	SENSITIVITY POTENTIOMETER (R13) – WIPER	CHECK_____
P2b-6	SENSITIVITY POTENTIOMETER (R13) –BOT	CHECK_____
P3b SIGNAL CONNECTOR		
P3b-1	SHELL VOLUME POTENTIOMETER (R30) – WIPER	CHECK_____
P3b-2	SHELL VOLUME POTENTIOMETER (R30) – TOP	CHECK_____
P3b-3	PITCH POTENTIOMETER (R29) – BOT	CHECK_____
P3b-4	PITCH POTENTIOMETER (R29) – WIPER	CHECK_____
P3b-5	SWEEP POTENTIOMETER (R15) – WIPER	CHECK_____
P3b-6	SWEEP POTENTIOMETER (R15) – TOP	CHECK_____
P4b SIGNAL CONNECTOR		
P4b-1	MASTER VOLUME POTENTIOMETER (R16) – TOP	CHECK_____
P4b-2	RANGE POTENTIOMETER (R28) – WIPER	CHECK_____
P4b-3	IMPACT VOLUME POTENTIOMETER (R31) – WIPER	CHECK_____
P4b-4	IMPACT TONE POTENTIOMETER (R14) – BOT	CHECK_____
P4b-5	UNUSED	
P4b-6	IMPACT TONE POTENTIOMETER (R14) – TOP	CHECK_____

Table 10 Module "B" Panel Wiring Checklist

MODULE "C" CONNECTOR WIRING CHECKLIST		
CONNECTOR REFERENCE	SIGNAL NAME & CONNECTION	CHECK IF INSTALLED
P2c SIGNAL CONNECTOR		
P2c-1	DECAY POTENTIOMETER (R33) – TOP	CHECK_____
P2c-2	TRIGGER J1 – TIP LUG	CHECK_____
P2c-3	STATUS LED CATHODE (-)	CHECK_____
P2c-4	SENSITIVITY POTENTIOMETER (R13) –TOP	CHECK_____
P2c-5	SENSITIVITY POTENTIOMETER (R13) – WIPER	CHECK_____
P2c-6	SENSITIVITY POTENTIOMETER (R13) –BOT	CHECK_____
P3c SIGNAL CONNECTOR		
P3c-1	SHELL VOLUME POTENTIOMETER (R30) – WIPER	CHECK_____
P3c-2	SHELL VOLUME POTENTIOMETER (R30) – TOP	CHECK_____
P3c-3	PITCH POTENTIOMETER (R29) – BOT	CHECK_____
P3c-4	PITCH POTENTIOMETER (R29) – WIPER	CHECK_____
P3c-5	SWEEP POTENTIOMETER (R15) – WIPER	CHECK_____
P3c-6	SWEEP POTENTIOMETER (R15) – TOP	CHECK_____
P4c SIGNAL CONNECTOR		
P4c-1	MASTER VOLUME POTENTIOMETER (R16) – TOP	CHECK_____
P4c-2	RANGE POTENTIOMETER (R28) – WIPER	CHECK_____
P4c-3	IMPACT VOLUME POTENTIOMETER (R31) – WIPER	CHECK_____
P4c-4	IMPACT TONE POTENTIOMETER (R14) – BOT	CHECK_____
P4c-5	UNUSED	
P4c-6	IMPACT TONE POTENTIOMETER (R14) – TOP	CHECK_____

Table 11 Module "C" Panel Wiring Checklist

MODULE "D" CONNECTOR WIRING CHECKLIST		
CONNECTOR REFERENCE	*SIGNAL NAME & CONNECTION*	*CHECK IF INSTALLED*
P2d SIGNAL CONNECTOR		
P2d-1	**DECAY POTENTIOMETER (R33) – TOP**	**CHECK_____**
P2d-2	**TRIGGER J1 – TIP LUG**	**CHECK_____**
P2d-3	**STATUS LED CATHODE (-)**	**CHECK_____**
P2d-4	**SENSITIVITY POTENTIOMETER (R13) –TOP**	**CHECK_____**
P2d-5	**SENSITIVITY POTENTIOMETER (R13) – WIPER**	**CHECK_____**
P2d-6	**SENSITIVITY POTENTIOMETER (R13) –BOT**	**CHECK_____**
P3d SIGNAL CONNECTOR		
P3d-1	**SHELL VOLUME POTENTIOMETER (R30) – WIPER**	**CHECK_____**
P3d-2	**SHELL VOLUME POTENTIOMETER (R30) – TOP**	**CHECK_____**
P3d-3	**PITCH POTENTIOMETER (R29) – BOT**	**CHECK_____**
P3d-4	**PITCH POTENTIOMETER (R29) – WIPER**	**CHECK_____**
P3d-5	**SWEEP POTENTIOMETER (R15) – WIPER**	**CHECK_____**
P3d-6	**SWEEP POTENTIOMETER (R15) – TOP**	**CHECK_____**
P4d SIGNAL CONNECTOR		
P4d-1	**MASTER VOLUME POTENTIOMETER (R16) – TOP**	**CHECK_____**
P4d-2	**RANGE POTENTIOMETER (R28) – WIPER**	**CHECK_____**
P4d-3	**IMPACT VOLUME POTENTIOMETER (R31) – WIPER**	**CHECK_____**
P4d-4	**IMPACT TONE POTENTIOMETER (R14) – BOT**	**CHECK_____**
P4d-5	**UNUSED**	
P4d-6	**IMPACT TONE POTENTIOMETER (R14) – TOP**	**CHECK_____**

Table 12 Module "D" Panel Wiring Checklist

J2 / J3 CHECKLIST		
CONNECTOR REFERENCE FROM	SIGNAL NAME & CONNECTION TO	CHECK IF INSTALLED
MODULE A		
J2-TIP TO:	**RANGE POTENTIOMETER (R28) – TOP**	**CHECK_____**
J3-TIP TO:	**MASTER VOLUME POTENTIOMETER (R16) –WIPER**	**CHECK_____**
MODULE B		
J2-TIP TO:	**RANGE POTENTIOMETER (R28) – TOP**	**CHECK_____**
J3-TIP TO:	**MASTER VOLUME POTENTIOMETER (R16) –WIPER**	**CHECK_____**
MODULE C		
J2-TIP TO:	**RANGE POTENTIOMETER (R28) – TOP**	**CHECK_____**
J3-TIP TO:	**MASTER VOLUME POTENTIOMETER (R16) –WIPER**	**CHECK_____**
MODULE D		
J2-TIP TO:	**RANGE POTENTIOMETER (R28) – TOP**	**CHECK_____**
J3-TIP TO:	**MASTER VOLUME POTENTIOMETER (R16) –WIPER**	**CHECK_____**

Table 13 J2/J3 Checklist

When you are all done wiring and you have checked every connection, then you can mount the board to its final position. Later on after you test everything; you can tie the wires together with wire ties to make things look neat and professional. This concludes the build phase of your Quad Bass++ … good luck !!!