Back

A Flexible Subwoofer Amplifier, Pt.2

The author explains how to make this project's printed circuit boards, which contain the "chips" of resistors that you will use to select the cutoff frequency f_{cLP} of the third-order Butterworth low-pass active filter.

By Rudy Godmaire

To explain the making of the PCBs, I will follow the same path presented in Part 1, which was guided by the block diagram (**Fig. 1**). All the circuitry is implemented on four small PCBs, which were designed with the help of ExpressPCB's free software and later manufactured by the same company in the US (www.expresspcb.com). This company offers the DIYer a very interesting alternative to home-brewed PCBs.

Not being well equipped to produce high-quality PCBs, I decided to give it a try. Within four days following my online order, UPS delivered the boards at my door . . . in Canada! Suffice it to say that I was truly impressed and satisfied with both the quality of the product and the service.

HEAT THE IRON

Stuffing a PCB is always a moment of great pleasure for me. I particularly enjoy seeing the parts take their place on the board. The different shapes and colors of the components produce a kind of artwork I like very much. Designing them is equally satisfying and much more challenging, as you will see.

The first three blocks of the diagram represent what you could call the preamplifier stage. My first prototype had all the components of this stage soldered on a double-sided board that was positioned vertically inside the amplifier. The complexity of the low-pass network forced me to run traces on both sides of the PCB, depriving me of the possibility to implement a ground plane.

The star ground point concept was nevertheless implemented by running multiple wires from this board to the single ground point of the amplifier. The design appeared to be nice but I soon realized that it was suffering from an unacceptably high level of noise. Measured at the binding posts, this noise varied between 3 and 5mV. Looking at the silver lining, the amplifier seemed to have no hum at all, which proved that the single ground point was doing its job.

I first thought the problem was due to thermal noise coming from the voltage regulators. Replacing the regulated power-supply board with two 9V batteries did not change a thing. I then started to investigate other possibilities; I changed a few capacitors, shielded the circuit from the toroidal transformer, and replaced the zener diodes and the op amps.

The situation remained the same despite these actions. However, in the process I noted that

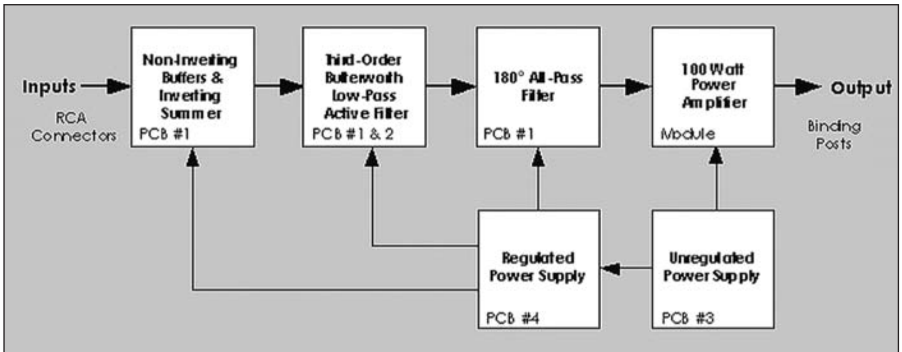


FIGURE 1: Block diagram of the subwoofer amp.

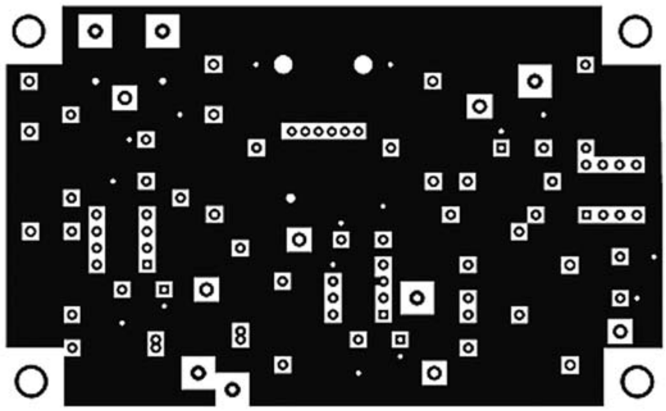


FIGURE 2: PCB #1 top copper layer (2.5″ × 4.0″).

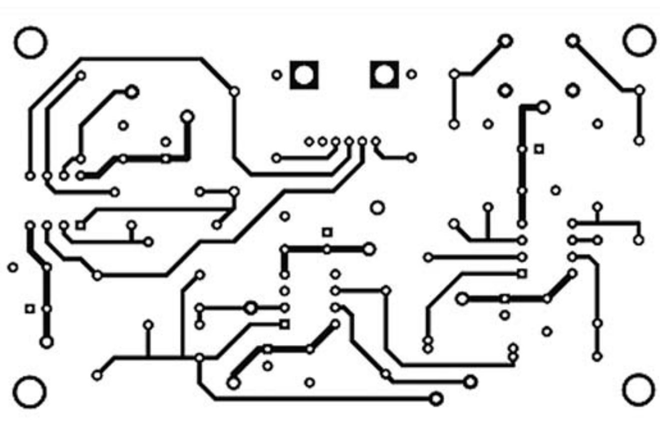


FIGURE 3: PCB #1 bottom copper layer (2.5″ × 4.0″).

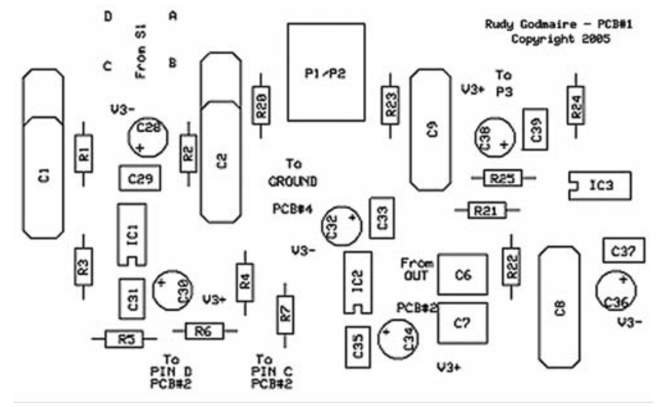


FIGURE 4: PCB #1 silkscreen layer (2.5″ × 4.0″).

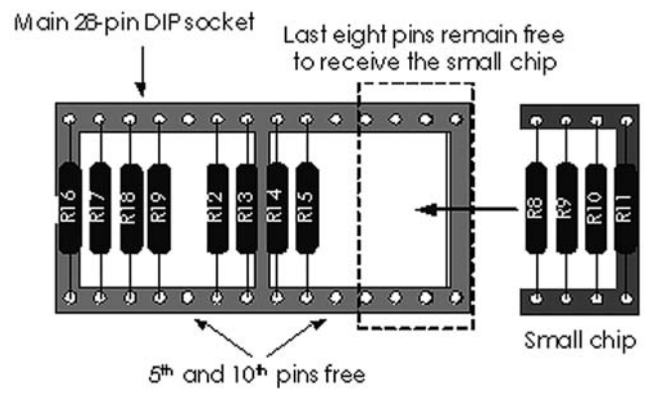


FIGURE 5: Soldering the resistors on the 28-pin DIP socket.

moving the wires returning to the star ground point and touching certain components had the effect of increasing or decreasing the noise. The problem was very difficult to diagnose and further study was required for me to understand what was going on.

Op Amp Applications[3] shed some light on my problem. "Section 7-2: PCB Design Issues" provides in-depth information about the conception of PCBs. Dos and don'ts are well explained. Though I was not completely sure about the source of the problem, I came to the conclusion that it had to be related to a bad circuit design. Back to square one!

Following the guidelines suggested in the book, I redesigned the entire circuit. To keep the traces as short as possible, I decided that two small circuits would be advisable. I implemented a ground plane on the board containing the three op amps (PCB #1).

On PCB #2—the one receiving S_2 and most of the network surrounding the low-pass filter—I had no choice but to allow the use of traces on both sides. I nevertheless implemented a partial ground plane. These efforts paid off; the hum and noise fell to 1.6mV. Using shielded interconnect wires in the signal path improved the situation further by decreasing it to 1.0mV at full level and 0.2mV at listening level.

PCB #1 & 2

NON-INVERTING BUFFERS &
INVERTING SUMMER
THIRD-ORDER BUTTERWORTH
LOW-PASS ACTIVE FILTER
VARIABLE 180° ALL-PASS ACTIVE FILTER

Stuffing PCB# 1 (**Figs. 2-4**) is very easy. I suggest that you solder the resistors first, because you might find it too tight in some places to place the resistors after the polypropylene capacitors and the stereo potentiometer.

Looking closely at C_1 and C_2, you will notice that you can solder capacitors having two different lead spacings. This will give you the flexibility to select a lower value than 0.47µF if you wish.

If you use eight-pin DIP sockets, as I did, I recommend that you wait before you insert your OPA-2604 in them. These are sensitive devices, so you should wait until you install everything in the chassis before proceeding. You might also want to power your amplifier for the first time without the op amps in place. This would give you the opportunity to adjust the voltages at the output of the voltage regulators (IC_4/IC_5) to make sure everything is fine.

Before you start soldering components on PCB# 2 (**Figs. 7-9**), prepare the "chips" of resistors (R_8 to R_{19}) that will determine the three cutoff frequencies of the subwoofer amp. To do so, use 28-pin DIP sockets.

Solder resistors R_{12} to R_{19} on one of these 28-pin DIP sockets (**Fig. 5**). Note that the fifth and tenth pins of each row will remain free. The same applies to the last eight pins on the right, which will further receive a smaller chip of resistors (R_8 to R_{11}), the ones that will provide you with the possibility to manually fine-tune the cutoff frequency for music listening.

You must use ¼W resistors because they fit nicely in the DIP sockets. I lean toward the excellent Holcos (the former non-inductive version), but other brands will do. In order to minimize the capacitance between the resistors, I soldered the resistors at two different heights. I soldered R_{12}, R_{14}, R_{16}, and R_{18} close to the surface of the socket while all other resistors were elevated by about ¼″.

You now need to prepare the small chips of resistors. The idea is to cut one or more 28-pin DIP sockets in three parts (**Fig. 6**). Once this is done, select the appropriate resistor values from *Table 1* and equation 3 of Part 1.

It is important to mention that the cutoff frequency for music will be in the vicinity of that of the main loudspeakers,

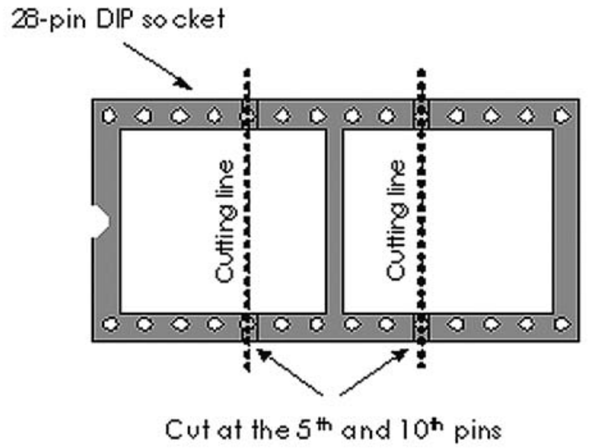


FIGURE 6: Cutting the 28-pin DIP socket to make three small chips.

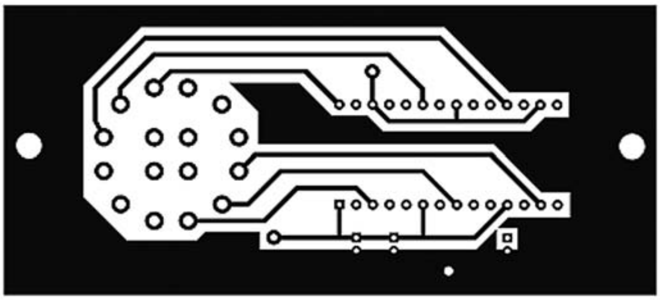


FIGURE 7: PCB #2 front copper layer (1.8″ × 4.0″).

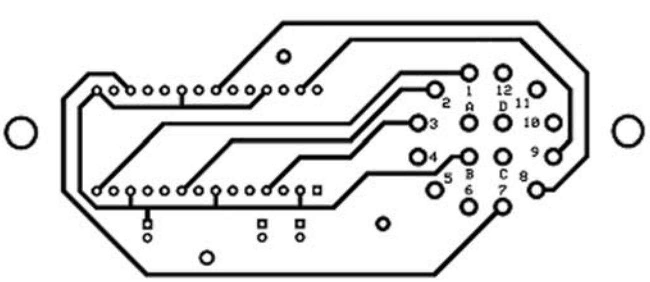


FIGURE 8: PCB #2 rear copper layer (1.8″ × 4.0″).

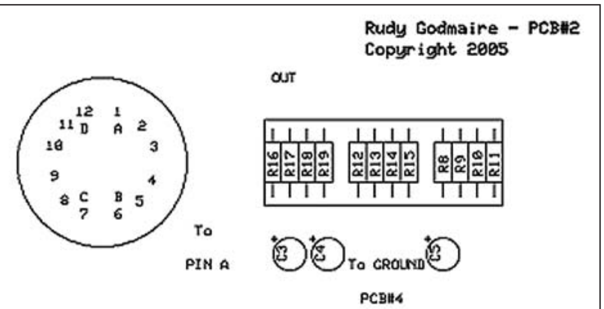


FIGURE 9: PCB #2 silkscreen layer (2.5″ × 4.0″).

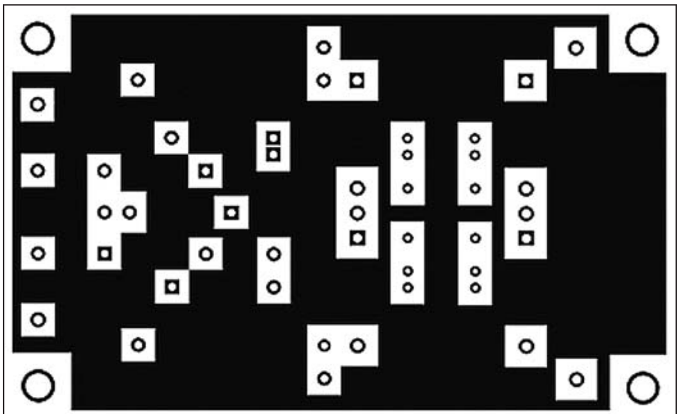


FIGURE 10: PCB #3 top copper layer (2.5″ × 4.0″).

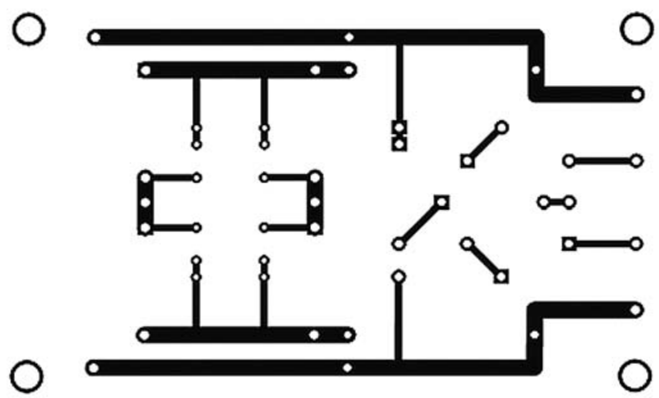


FIGURE 11: PCB #3 bottom copper layer (2.5″ × 4.0″).

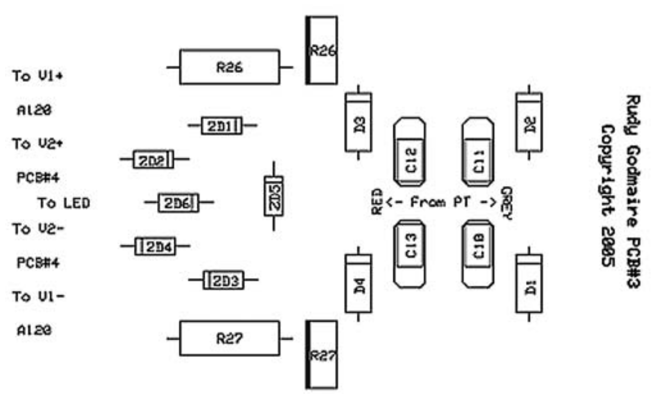


FIGURE 12: PCB #3 silkscreen layer (2.5″ × 4.0″).

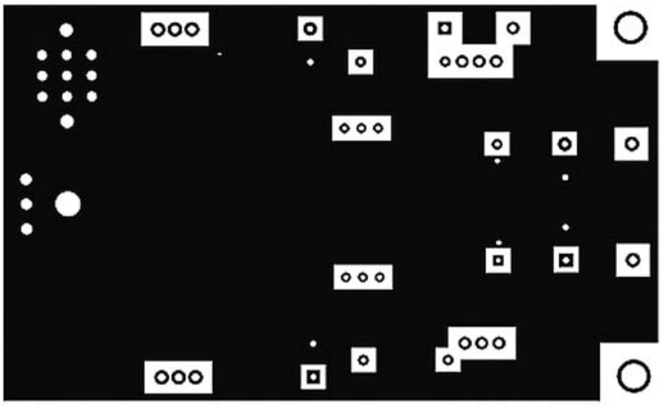


FIGURE 13: PCB #4 top copper layer (2.5″ × 4.0″).

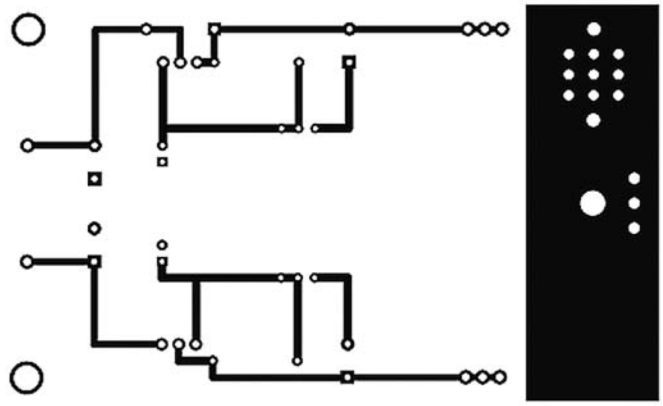


FIGURE 14: PCB #4 bottom copper layer (2.5″ × 4.0″).

40Hz in my case. When preparing these chips, it is a good idea to select different values around that frequency. This is why I prepared five chips of resistors that provided me with the following cutoff frequencies: 35Hz, 39Hz, 44Hz, 48Hz, and 52Hz.

Simply swapping these chips in the amplifier enabled me to fine-tune the integration of my subwoofer in a very convenient manner. As I will explain in Part 4, I ended up with a cutoff frequency of 48Hz. Experimentation will tell you which value best suits your system. When soldering the resistors to the chips, I elevated R_8 and R_{10} by ¼″ for the same reason.

The orientation of the chip is very important when you plug it into the main 28-pin DIP socket, because R_9 is twice the value of R_8, R_{10}, and R_{11}. Putting a small label on top of the chip stating the cutoff frequency will offer the dual advantage of identifying the chip and helping you orient it the right way.

The array of resistors will be related to S_2 in the following sequence:

- Position 1: R_8 to R_{11} (f_{cLP} = 48Hz)
- Position 2: R_{12} to R_{15} (f_{cLP} = 80Hz)
- Position 3: R_{16} to R_{19} (f_{cLP} = 100Hz)

You are now ready to solder the components to the PCB. The first component to put in place is the frequency selector S_2, which, along with all the other parts, you must install on the ground plane side. You might need to slightly bend some pins of S_2 to insert it. Just proceed gently and everything will be fine.

Before you solder the switch, I strongly suggest that you perform a conductivity test with a DMM on each and every position. You do not want to discover that your switch is defective after it has been soldered to the board because it will be very annoying to try to remove it from the PCB. Take my word; do not skip this quality-control step. Also, make sure you do not overheat the pins; otherwise, the plastic might have a tendency to melt around them.

In my case, soldering S_2 has been eased by the fact that my PCBs have through-hole plating. This enabled me to apply solder from the backside of the board. If you make your own PCBs, you will need to solder some of the pins from the front side (pins #4, 5, 6, 10, 11, 12). This might offer you a little challenge, but you should not have any problem if you act carefully.

When you are done with S_2, you may proceed with the main 28-pin DIP socket and the rest of the components. Pay attention to the orientation of the socket; when looking at the PCB on the

FIGURE 15: PCB #4 silkscreen layer (2.5″ × 4.0″).

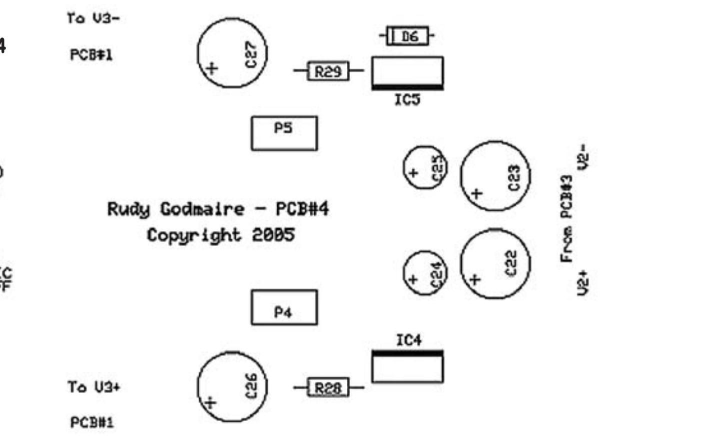


TABLE 1: PARTS LIST OF THE PCBS.

Component/Quantity	Description	Manufacturer	Supplier	Part No.
PCB # 1				
R1-4	100kΩ/¼W resistors	Holco H8*	PC	HOLCO-53921
R5-7, R21, 22, 45, 25	56kΩ/¼W resistors	Holco H8*	PC	HOLCO-50046
R20, R23	5kΩ/¼W resistors	Holco H8*	PC	HOLCO-64731
C1-2	0.47μF/160V capacitors	Wima MKP-10	WL	WM219
C6-7	0.022μF/63V capacitors	Wima FKP-2	WL	WM14
C8-9	0.1μF/100V capacitors	Wima MKP-10	WL	WM209
C28, 30, 32, 34, 36, 38	10μF/16V capacitors	Black Gate	MP	**
C29, 31, 33, 35, 37, 39	0.01μF/63V capacitors	Wima FPK-2	WL	WM13
P1/P2	50kΩ stereo potentiometer	Panasonic	DK	P2A5503-ND
IC1-3	dual op amps (OPA2604)	Burr-Brown	DK	OPA2604AP-ND
3	8-pin DIP sockets	Mill-Max	DK	ED3108-ND
PCB # 2				
R8-11	Resistor chips-see text	Holco H8*	PC	HOLCO-XXXXX
R12, 14, 15	8k9Ω/¼W resistors	Holco H8*	PC	HOLCO-53840
R13	17k8Ω/¼W resistors	Holco H8*	PC	HOLCO-53864
R16, 18, 19	7k1Ω/¼W resistors	Holco H8*	PC	HOLCO-53832
R17	14k2Ω/¼W resistors	Holco H8*	PC	HOLCO-53858
C3	0.22μF/50V capacitors	Black Gate PK	MP	**
C4	0.33μF/50V capacitors	Black Gate PK	MP	**
C5	0.47μF/50V capacitors	Black Gate PK	MP	**
S2	4P3T rotary switch	C&K	DK	CKC7007-ND
PCB # 3				
R26-7	1R0/2W resistors	Caddock MP-930	PC	CADDOCK-51128
D1-4	3.3A/100V	Schottky diodes Inter. rectifier	DK	31DQ10-ND
C10-13	0.01μF/250V capacitors	Wima MKP-10	WL	WM201
ZD1-5	16V/5W zener diodes	Microsemi	DK	1N5353BMSCT-ND
ZD6	30V/5W zener diodes	Microsemi	DK	1N5363BMSCT-ND
3	28-pin DIP sockets	Mill-Max	DK	ED3728-ND
PCB # 4				
R28-9	100R/¼W resistors	Holco H8*	PC	HOLCO-54533
P4-5	1kΩ 25 turns potentiometer	Bourns	DK	3296X-102-ND
C22-3	10μF/50V capacitors	Black Gate	MP	**
C24-5	10μF/16V capacitors	Black Gate	MP	**
C26-7	220μF/16V capacitors	Black Gate	MP	**
D6	1A/100V diode	Microsemi	DK	1N4002MSCT-ND
IC4	voltage regulator (LT1085CT)	Linear Technology	MP	**
IC5	voltage regulator (LT1033CT)	Linear Technology	MP	**
2	T0-220 5W clip-on heatsinks	AAVID Thermalloy	DK	HS119-ND

Digi-Key: www.digi-key.com
Parts Connexion: www.partsconnexion.com
ExpressPCB: www.expresspcb.com
Michael Percy: www.percyaudio.com
Welborne Labs: www.welbornelabs.com

* These are the new Holcos. Unfortunately, the former non-inductive version is not manufactured anymore.
** Michael Percy does not use part numbers. Consult his catalog for more details.

groundplane side, R_{16} must be the closest resistor to S_2.

PCB #3

UNREGULATED POWER SUPPLY

The last two PCBs concern the power supply of the amplifier. The purpose of PCB #3 (**Figs. 10-12**) is to transform the AC voltages coming from the secondary winding of the power transformer (33V AC) to DC voltages (±48V DC and ±16V DC).

Stuffing this PCB is very straightforward. You just need to pay attention to the orientation of the Schottky diodes and the zener diodes.

If you take a closer look at the bypass capacitors (C_{10} to C_{13}), you will realize that I have designed the board so it can accommodate capacitors having two different lead spacings. Depending on the type of polypropylene capacitors you purchase, you will use one position or the other. The same applies to the power resistors (R_{26}/R_{27}). The configuration offers you the possibility to solder T0-220 type resistors as I did, or to use radial types.

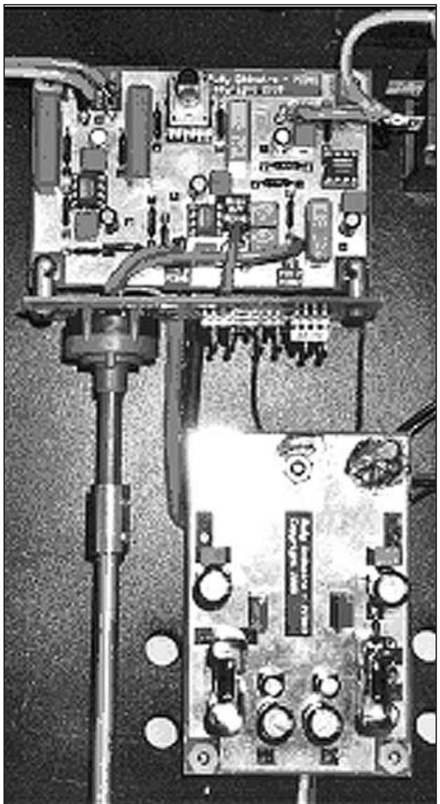
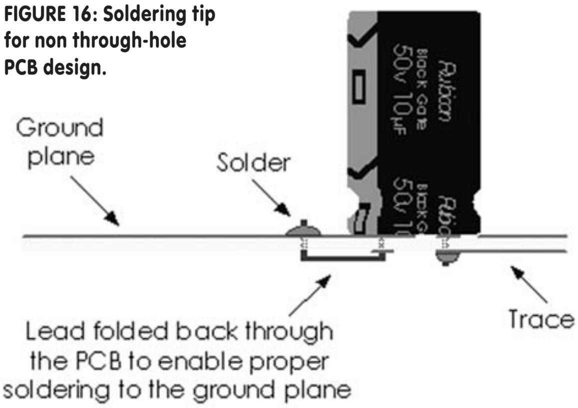


PHOTO 1: PCBs 1, 2, and 4 fully stuffed and installed.

FIGURE 16: Soldering tip for non through-hole PCB design.



PCB #4

REGULATED POWER SUPPLY

The main objective of PCB #4 (**Figs. 13-15**) is to provide clean DC voltages to the op amps located on PCB #1. Bad regulation would translate into undesirable hum at the speaker end. Another advantage of this power supply stage is that it offers a means to adjust the intensity of these voltages. Experimentation showed me that output voltages of ±8V DC give excellent results in my application in terms of noise.

Again, I suggest that you proceed with the smaller components and then move on with the larger ones, keeping the ICs and their 5W heatsinks for the end. If you pay attention to the design of the PCB, you will notice the implementation of the recommendations of Linear Technology discussed in Part 1 concerning the connection of R_{28}/R_{29} and P_4/P_5 to ensure the best load regulation possible.

Note also that PCB #4 has a large ground area on the soldering side. This section combines with the ground plane and both act as the star ground point for the entire amplifier. This means that all ground connections gather at this place. This star ground point is also connected to the chassis by means of the third standoff of this PCB. So make sure you use a metallic standoff!

When assembling the PCB inside the preamplifier, remember to connect the ground planes of PCB #1 and 2 to the star ground point. Using 16 AWG wire, you will need to connect the ground plane of these PCBs to the ground point of PCB #4.

The PCBs I have just described benefited from a through-hole plating design, but not if you make your own PCBs from the preceding figures. Consequently, this will force you to solder a certain number of components from the top copper layer.

You should not have any problem except with the capacitors having a lead soldered to the ground plane. A very easy way to deal with this issue is shown in **Fig. 16**. Drilling a small hole beside the capacitor and folding back the lead of the capacitor through the PCB will do the trick.

CONCLUSION

This second part of the article has focused on the making of the four PCBs and the resistor chips. In Part 3, I will address all the aspects of the construction of the subwoofer amp. *aX*