By Thomas M. Perazella

Maintaining Your Balance, Part 2

Follow this author's instructions and you're on your way to better LP-to-CD conversions.

LOCATING COMPONENTS ON THE BOARD

Using the program ExpressPCB, first determine the dimensions of the board based on components' sizes and shapes. Once that is set into the layout program, placement of the components can begin, the easiest way being to use the component manager to select a component outline and place it into the layout. **Photo 1** shows an electrolytic capacitor with 0.2″ lead spacing.

Note that if the component is polarized, the positive lead has a square pad, a convention that makes correctly orienting the component easier. You can also rotate the component before insertion into the layout. Once it is in the layout, you can move the component on the grid and rotate it to different orientations. You will need to make sure that the placement allows for the maximum component dimensions so that it does not interfere with adjacent components. The grid markings make that simpler. Be especially careful of items such as heatsinks, which can extend well beyond

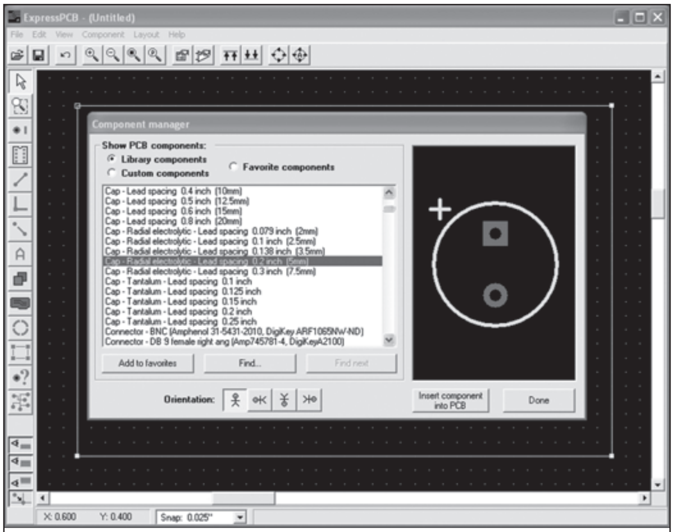


PHOTO 1: Device in Component Manager.

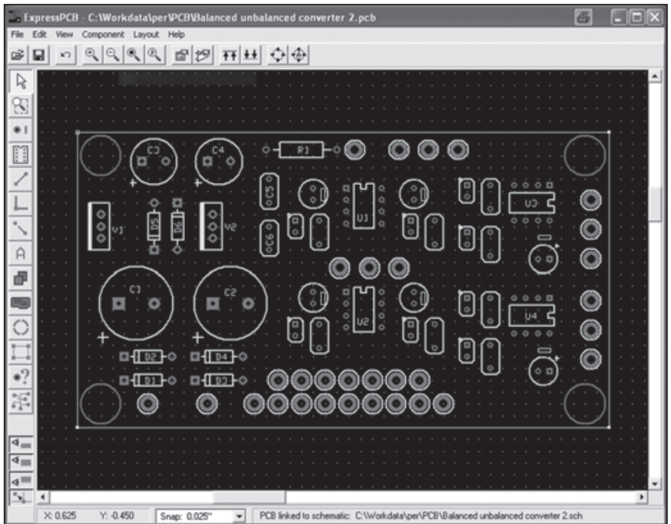


PHOTO 3: Parts layout.

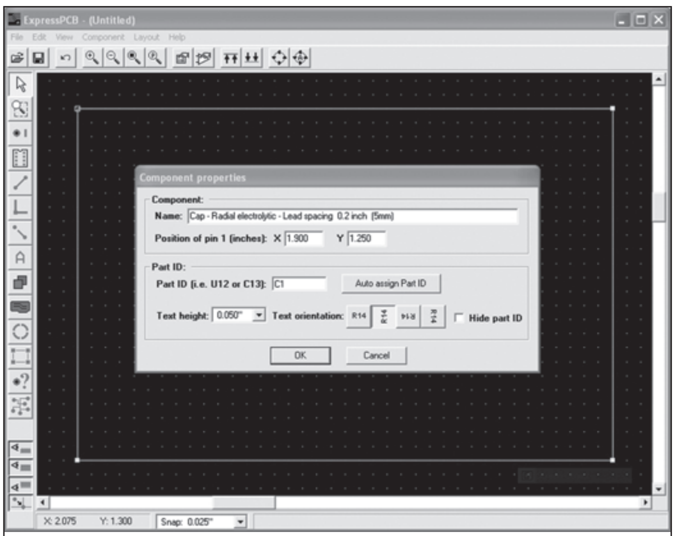


PHOTO 2: Setting component properties.

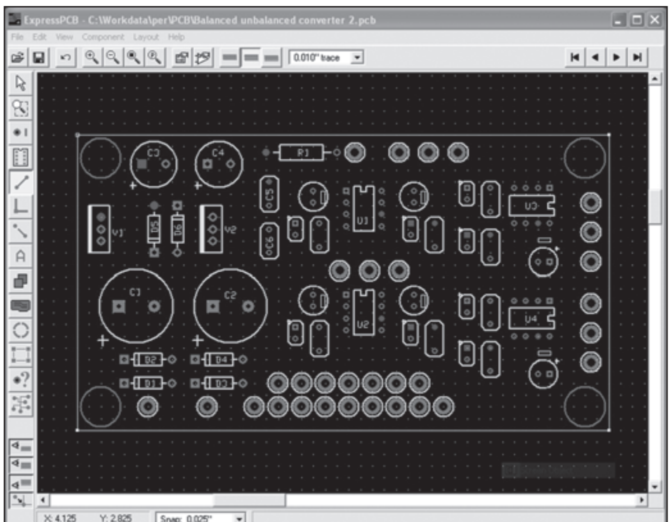


PHOTO 4: Highlighted pins.

the device outline.

If you have previously produced a schematic for this circuit using ExpressSCH, you can achieve other results at this time. By linking the board to the schematic, you can set device properties as shown in **Photo 2**.

Once linked, the board program knows all the connections. If you set the position of pin 1 in the horizontal and vertical positions, you will not need to drag it into position once in the layout. You can also choose the designation from the schematic, which will assist in the trace layout. The component will automatically assign the text used in the schematic to the component in the layout. Sizing and rotation of the text is also possible.

LAYING DOWN TRACES

After the components are in place, you need to lay down the traces to connect the components. This is where generating a schematic in ExpressSCH can be a major time-saver, because ExpressPCB will automatically indicate the pins that you need to connect. **Photo 3** shows the parts layout of the project board before the traces were defined.

To begin the trace layout, you can click on any pad and all the pads that are connected together on the schematic will be indicated by a color change to blue. In **Photo 4** the output of V1 was highlighted and the following pins have been marked in blue:

The output of V1
The anode of D5
The positive side of C3
One side of R1
One side of C5
Pin 6 of U1
One side of each B+ power-supply bypass capacitor
Pin 7 of U3
Pin 7 of U4
Pin 6 of U2.

You can now use the trace layout tool to connect the highlighted pins with appropriately sized traces. You can also pick which side of the board (top or bottom) a particular section of trace should use. For example, you may choose to have a connection on the top of the board go to one pin and have the connection from that pin go to the next component on the bottom of the board. The plated-throughhole will connect both sides of the board. This helps reduce the need for jumpers.

Once you have made all the connections to a particular set of interconnected pins, you can highlight another pin from a different group and repeat the process until all the traces are laid. The completed trace map is shown in **Photo 5**.

Both programs include many more tools which are not discussed here. The best way to learn about them is to download the programs and experiment.

You are now ready for final checking before sending the file out for board production. To assist in checking, the program provides several options to print various board configurations as shown in **Photo 6**.

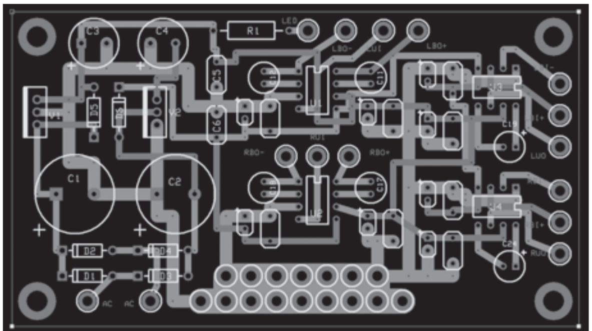


PHOTO 5: Completed trace map.

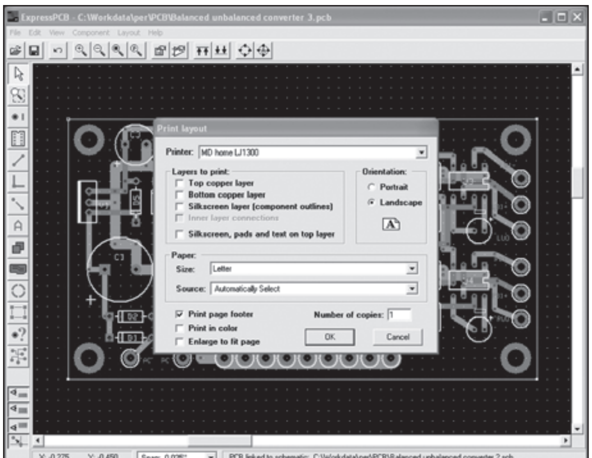


PHOTO 6: Board print choices.

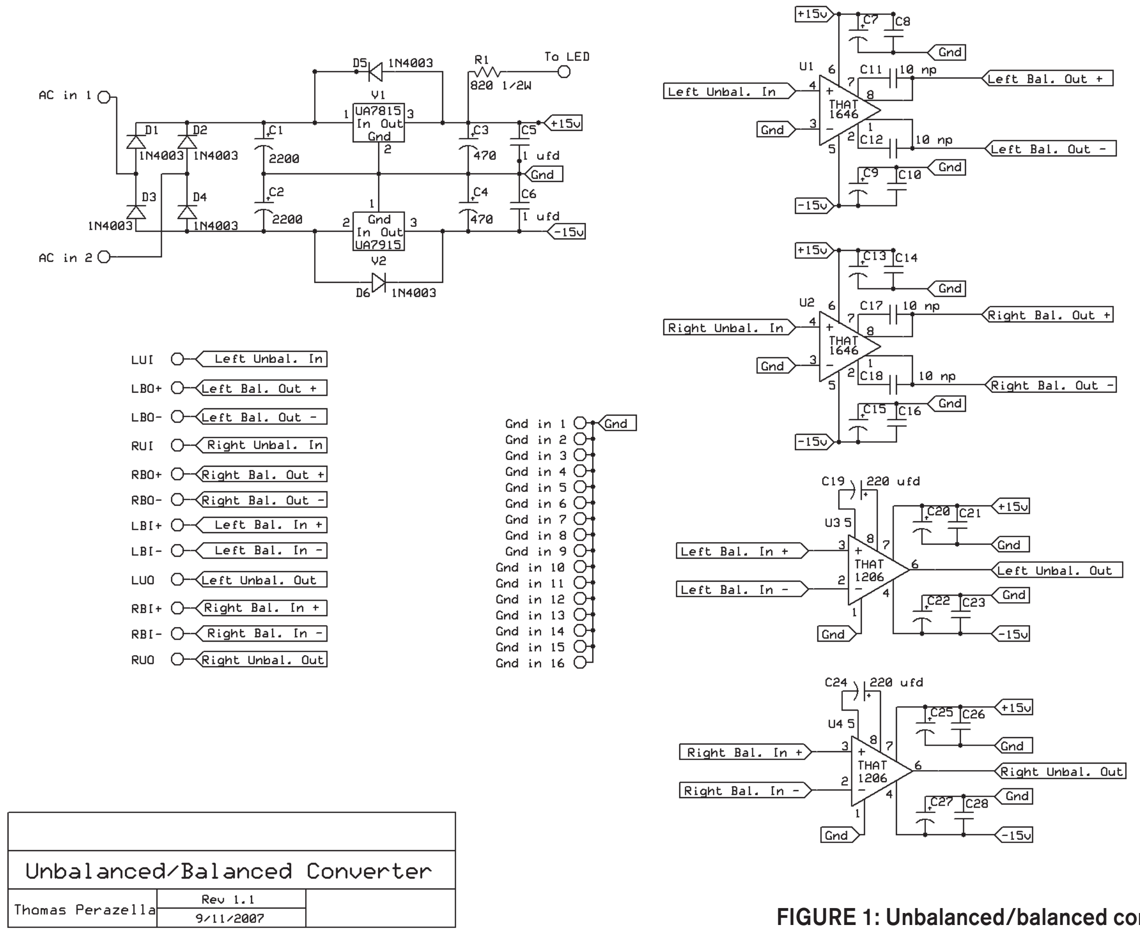


FIGURE 1: Unbalanced/balanced converter circuit.

You can print out several aids including:

- A schematic as previously shown in **Fig. 1**.
- A component layout diagram without pins as shown in **Fig. 2**.
- A component layout diagram with pins as shown in **Fig. 3**.
- A trace map of the top side of the board as shown in **Fig. 4**.
- A trace map of the bottom side of the board as shown in **Fig. 5**.

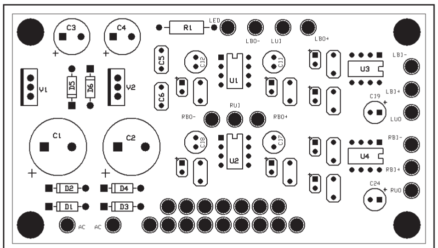


FIGURE 2: Layout diagram.

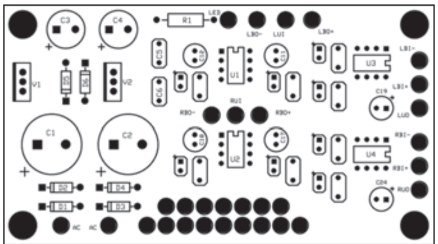


FIGURE 3: Components with pins and text.

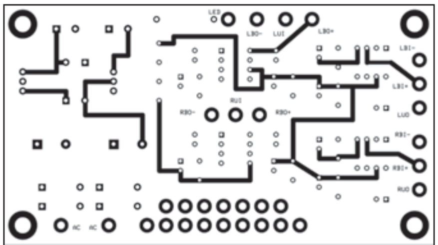


FIGURE 4: Top traces.

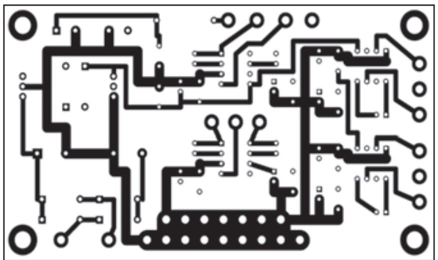


FIGURE 5: Bottom traces.

Using these, you can trace all connections to ensure the file is correct before sending it out for processing. One thing is certain—complacency leads to errors. I checked the schematics and maps several times before sending the file out, and everything checked OK. However, I was complacent and checked the pin connections for the positive regulator from memory because I had used that device so many times. I remembered the pinouts incorrectly. As a result, I reversed the output and ground connections, which went unnoticed until the boards were built and I began testing. Shame on me!

I sent out the file via e-mail, and the completed boards arrived a few days later. The quality looked very high. Views of the top and bottom of the boards are shown in **Photos 7** and **8**.

As is my regular process for building circuits, I build the power supply sections first and test them before adding other components. This helps prevent catastrophic failure in case the power supply has problems.

After the power supply components were added to the board, I connected the transformer and slowly increased the input voltage with a Variac while measuring output voltage. The mistake I made with the regulator pin connections was immediately evident in the output voltage measurements. After checking the board and finding that it was correct to my specifications, I looked up the pin diagram for the regulator and realized my mistake. I made a simple manual fix to the board to re-route the traces. Pictures of the fixes for both sides of the board are shown in **Photos 9** and **10**.

I removed two of the traces with a sharp knife blade and used replacement connections made out of solid hook-up wire. The re-tested supply worked properly. I re-checked my other connections and corrected the schematic and board layouts—all the diagrams in this article are correct. I added the rest of the components to the board and tested the completed board (**Photo 11**).

Preparation of the housing started with selection of the appropriately sized die-cast aluminum box. I prefer metal boxes to plastic for their ease of machining and shielding. I mounted the transformer and circuit board on the base of the box with all the signal connectors attached to the main body of the box, and with all the necessary holes drilled and de-burred. I then cleaned, primed, and painted the box parts.

In the past, when doing a one-off project, I would use press-type letters with a protective coating of clear urethane. Unfortunately, those type sheets are no longer to be found. For this project, I formed the letters and symbols using a Brother P-Touch label printer with a tape cartridge having white type on a clear background. It worked, but was not as professional-looking as the press-type. The finished box is shown in **Photo 12**.

To mount the board and transformer, I made two sets of brackets from flat steel barstock. The brackets allowed vertical mounting of the two pieces for better internal space utilization. I wired the transformer to the board as shown in **Photo 13**. Note the central chassis grounding lug mounted under the nut on one of the circuit board brackets.

The next step was to mount the signal connectors to the box. One of the biggest problems with noise in balanced circuits is the so-called "pin-1 problem" as detailed by Neil Muncy in his June 1995 paper. The preferred method of grounding XLR connectors is to connect pin-1 and any connector-shell grounds to the chassis as close to the connector as possible. **Photo 14** shows the pin-1 con-

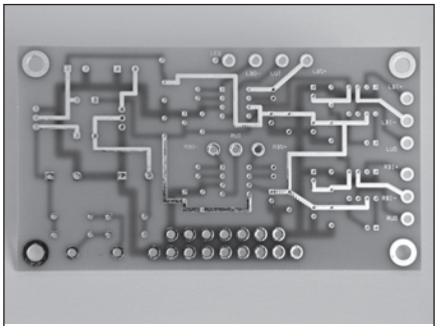


PHOTO 7: Top of board.

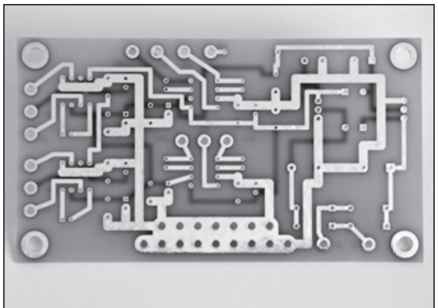


PHOTO 8: Bottom of board.

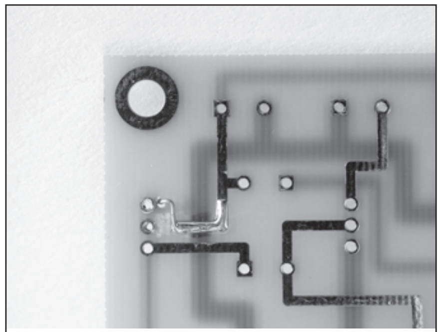


PHOTO 9: Fix for error on top side of board.

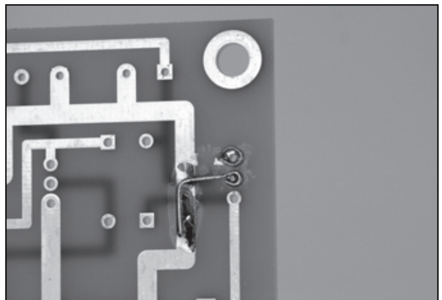


PHOTO 10: Fix for error on bottom side of board.

nections from the XLR connectors going to a chassis ground lug right next to the connectors.

Photo 15 shows the RCA connectors using insulators to float them from the chassis at the input point. Unlike the balanced XLR connectors, the shells of the RCA jacks are signal-carrying and are connected through individual wires to the central ground point on the circuit board rather than at the chassis. **Photo 16** shows the ground connections on the board as well as ground leads from the box and the power cord going to the chassis ground on the lid.

The wire connection from the ground lug on the box and the ground lug on the lid provides a ground connection when the lid is off for testing. Once closed, the mating sections of the metal box provide the definitive overall chassis ground. After making the power input and indicator connections, I was ready to test and close up the box. **Photo 17** shows the

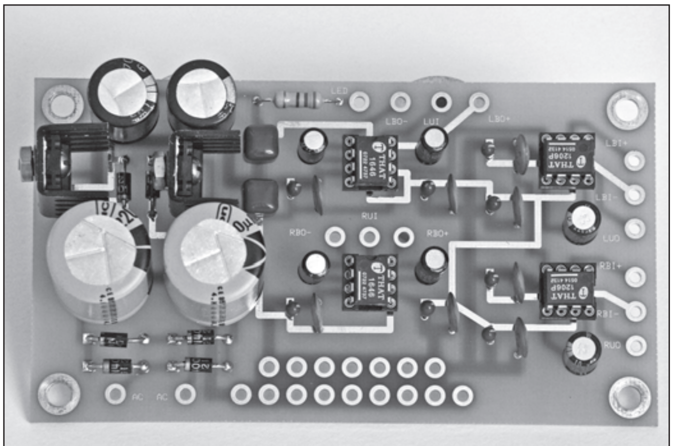


PHOTO 11: Completed PC board.

PHOTO 12: Completed box.

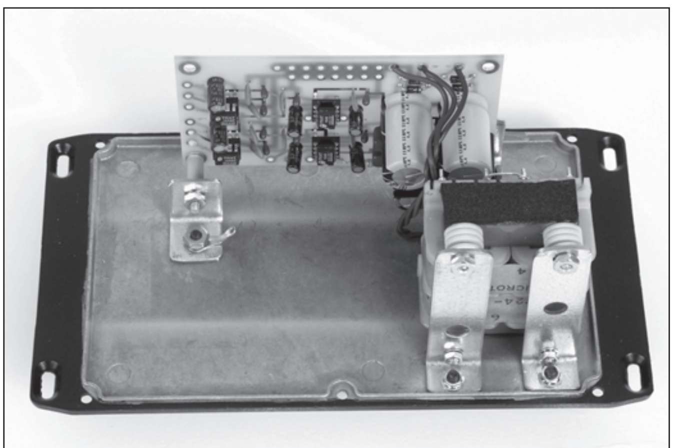


PHOTO 13: Transformer wiring.

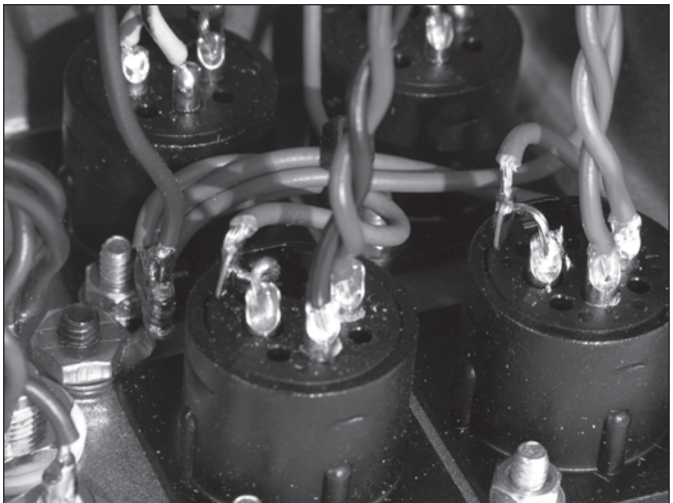
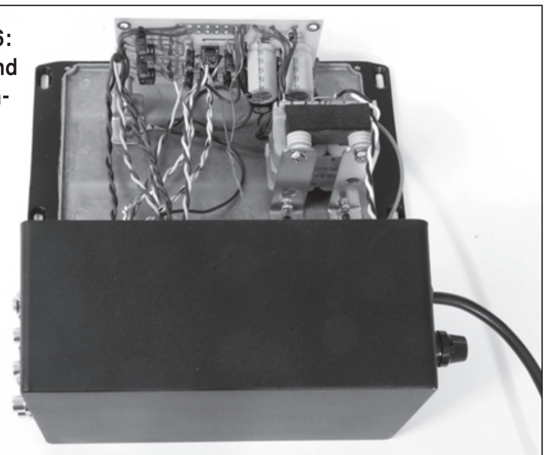


PHOTO 14: Pin 1 grounding.

PHOTO 15: RCA jacks insulated from chassis.

PHOTO 16: Ground and signal connections.



completed project just before closure.

Photos 18 and **19** show the completed box and a close-up of the connector arrangement. Note that for the unbalanced connectors there is one pair for input and three pairs for output. The triple output allows for connections to the computer sound card, a headphone amplifier, and any other test equipment I might choose to put into the circuit.

TESTING

A good measure of how well an electronic device performs is to determine how closely it compares to a straight wire when a signal is passed through. To test this project, I made two cables that would allow me to loop the output of the balanced circuit back into the balanced input. That would allow me to input a signal to the unbalanced circuit and measure the output signal of the unbalanced circuits after passing through two conversion steps. I used test sine waves of 1V RMS at 5Hz, 1kHz, and 500kHz with the input and output signals displayed on my oscilloscope.

Figures 6, **7**, and **8** show the results at 5Hz, 1kHz, and 500kHz, respectively, with the input signal as the top trace and the output signal as the bottom trace. They are virtually identical. To make

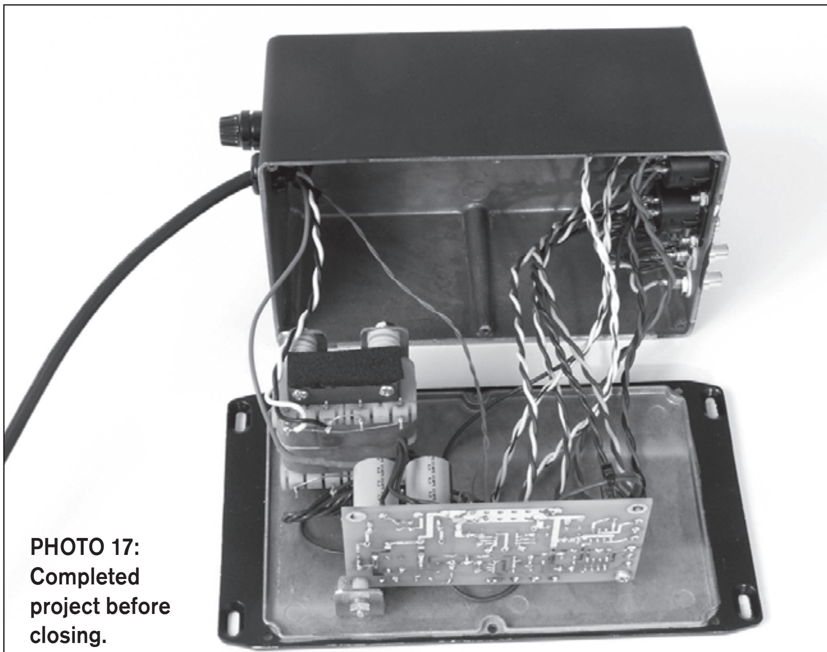


PHOTO 17: Completed project before closing.

PHOTO 18: Completed project.

the test harder, I repeated the test using square waves at the same frequencies. **Figures 9**, **10**, and **11** show the results with no visible problems.

For an even more difficult test, I increased the signal level to 3V RMS and expanded the trace of the 500kHz square wave to determine whether there were any differences, even though they would be far outside the audible band. **Figure 12** shows the leading edge of the square wave with a time-base setting of 100ns/division.

You can see a rise-time lag of about 100ns, with a very slight overshoot lasting about 100ns. This is certainly not audible and indicates the excellent performance of both devices.

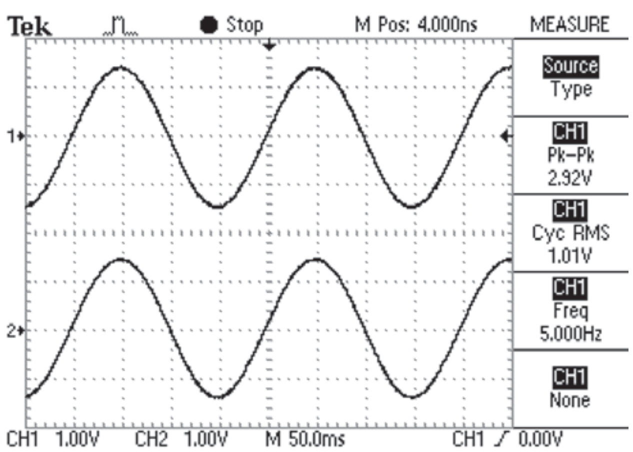


FIGURE 6: Sine wave performance at 5Hz.

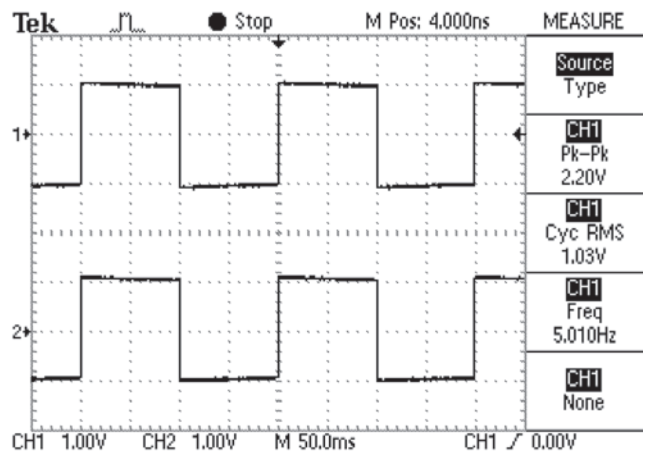


FIGURE 9: Square wave performance at 5Hz.

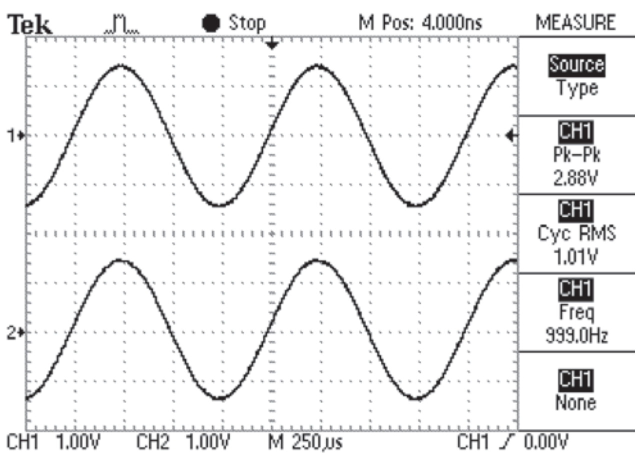


FIGURE 7: Sine wave performance at 1kHz.

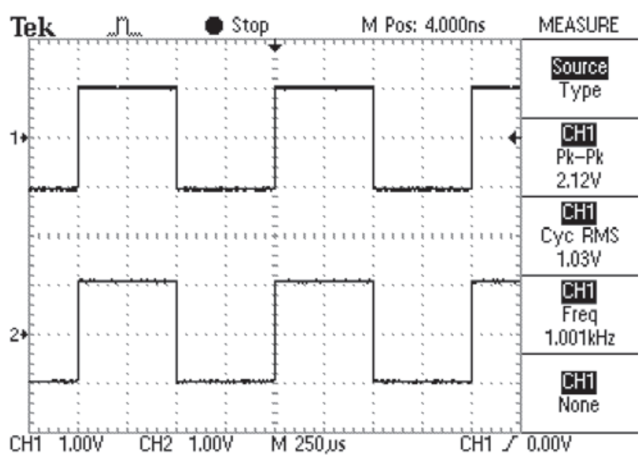


FIGURE 10: Square wave performance at 1kHz.

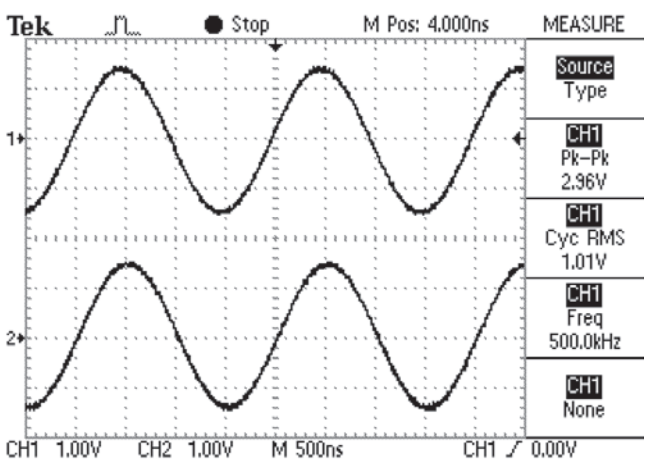


FIGURE 8: Sine wave performance at 500kHz.

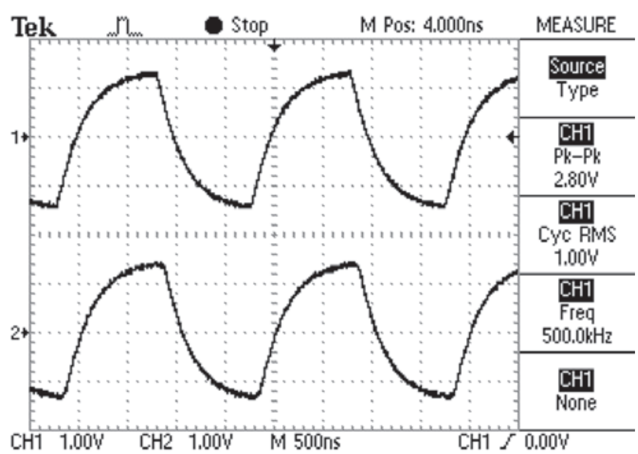


FIGURE 11: Square wave performance at 500kHz.

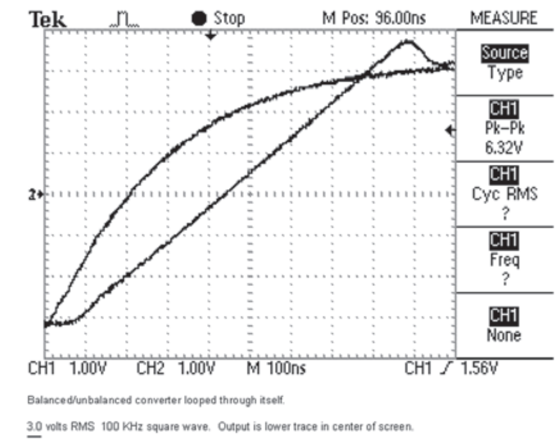


FIGURE 12: 3V square wave at 500kHz.

PHOTO 19: Connector arrangement.

I wanted a live test in a recording venue, and contacted my friend Michael Morgan,[8] who makes fine professional recordings for NPR and others. Although most of his equipment uses balanced interconnections, he set up a test using unbalanced and balanced connections. The project worked with no problems.

Finally, I connected it in my LP-to-CD conversion setup where it worked as hoped with no audible added noise or distortion. **Photo 20** shows that setup, along with my Dual turntable with Shure Ultra 500 cartridge, my home-built RIAA phono preamp based on OP27 op amps, the balanced-to-unbalanced converter, a Behringer DEQ2496 equalizer, a Headroom headphone amp, a pair of Etymotic ER4S earphones, and the output to my PC.

I have only had time to burn a few CDs, using Audacity™ as the recording software, but the results have been great.

CONCLUSIONS

The addition of the THAT Corporation balanced-to-unbalanced circuits has eliminated many of the problems inherent in those types of conversions without the expense and frequency response limitations of many available transformers. High-quality transformers still hold the top spot for ultimate CMRR at higher frequencies, and are mandatory where significant and possibly dangerous voltage offsets exist. However, for most situations, these THAT ICs will provide excellent performance that is certainly far beyond using adapter cables. Kudos to Bill Whitlock and the folks at THAT Corporation. *aX*

REFERENCE

8. Michael Morgan, Isle of Skye Audio Productions, www.RecordClassical.com.

PHOTO 20: Converter in the system.