NOTES & ERRATA FOR PROJECTS PUBLISHED IN SILICON CHIP (2010)

Please note: errata apply primarily to the print edition of SILICON CHIP as online issues are normally changed when an error is identified. However some errata may still apply to the online edition; check carefully before making any changes to a project.

Balanced Output Board For The Stereo DAC, January 2010: Pins 1 & 2 of the XLR sockets are shown transposed on the circuit diagram (Fig.1, page 43). The PC board and the parts layout diagram (Fig.2, page 44) are correct.

Note also that phantom power should not be applied to the XLR sockets of the Balanced Output Board (ie, phantom power should be switched off). Alternatively, cut the tracks between the 100 ohms resistors and the XLR sockets and install 10uF bipolar (BP) electrolytic capacitors across the gaps (ie, in series with the pin 2 & pin 3 outputs). (02/10)

PICAXE-Controlled Watering System, Circuit Notebook, February 2010: The suppression capacitor across the motor should be increased from 10nF to 220nF. (03/10)

Milliohm Meter Adaptor for DMMs, February 2010: The output reference pin of IC3 (pin 5) should not be connected to the Set Zero trimpot VR5 but should be connected to PC board ground instead. This will allow VR5 to set the output zero correctly. To make this change, the copper track currently connecting to pin 5 of IC3 should be cut and the pin connected to pin 4 of the same IC via a short length of hookup wire. To give VR5 a greater adjustment range in order to cope with a "worst case" situation, the resistor in parallel with VR5 can be changed from 100 ohms to 390 ohms. (10/10)

Digital Audio Signal Generator, March-May 2010: (1) The original software did not set up the crystal oscillator and PLL correctly. It worked on some dsPIC33 chips but not all. The updated source code and hex file has been provided to kit suppliers and is available on our website (0420310B.hex). (06/10)

(2) There is a bug in the firmware. If the default sample rate is changed from 48kHz (ie, settings are written to Bank 0 after the sample rate is changed) then the generator will fail to start up. The solution is to reflash the dsPIC microcontroller (IC4) with a new version of the firmware (0420310C.hex). This revised firmware can be downloaded from the ***SILICON CHIP*** website. (08/10)

Solar-Powered Lighting System, May & June 2010: the 2.2kΩ resistor shown above IC1 in the PCB overlay diagram (Fig.6) should be 22kΩ. (11/26)

Vintage Radio, June 2010: The primary of the IF transformer in Fig.6(b) on page 94 should be connected to earth (chassis) and not to HT as shown. The RF choke (RFC) goes to the HT line. (07/10)

Dual Tracking Power Supply, June-July 2010: (1) The labels for V± and Ilim were swapped on the front panel artwork. Updated front panel artwork has been provided to the kit suppliers and is now available on the ***SILICON CHIP*** website. In addition, some constructors have found that VR7 has insufficient range to properly trim the V± reading. In this case, increase the value of its 68 ohms shunt resistor or remove the shunt resistor entirely. (08/10)

(2) There are two errors in the circuit diagram of Fig.2 in the June issue. The 820 ohms resistor connected to pin 1 of IC4a should be 68 ohms while the 100k resistor from Vout- to the top of VR7 should be 10k. The layout diagram (Fig.8) in the July 2010 issue is correct. In addition, the wiring diagram (Fig.11) shows the top and bottom connections to the VOLT ADJ header (CON6) transposed. Also, the leads to pins 5 & 6 on the panel meter must be transposed for the Altronics display. (09/10)

(3) Under some conditions, if the supply is switched off then on again soon after, the negative output current sense display may not operate correctly. To solve this, a 1N4148 or 1N4004 diode can be soldered between the output of IC2a and ground, with the anode to ground. This can either be soldered to the underside of the main PC board or to the electrically connected pads on the rear of the front panel board. (01/11)

(4) The circuit diagram on pages 70-71 of the June 2010 issue shows the wiper of VR2 connected to one end of the track when it should not be. The circuit board and overlays are correct. In addition, the two wires from the "LED PWR" connector on the main board to the front panel board should be swapped on the wiring diagram on page 84, July 2010. Finally, the wiring for the Jaycar LED panel meter shown is correct according to the manufacturer's data sheet but we have found that the connections to its pin 7 & 8 (or pins 5 & 6 on the front panel board) must be swapped for it to work correctly. In practice, this simply involves swapping the yellow and red wires to the display connector. (08/11)

Remote Controlled Digital Up/Down Timer, August 2010: The relay contacts are incorrectly shown by the screened overlay on the PC board. The parts layout diagram of Fig.2 is correct. In addition, the LED display part numbers are incorrect. These should be 7DR8021BS and 7DR5621BS.

A couple of minor issues have also surfaced in the software. First, the alarm period is skipped if the external trigger is active when the countdown ends. Second, the device ignores infrared signals if the reset and trigger inputs are on

simultaneously. The updated firmware (1910810B.hex) fixes both issues and has been forwarded to the kit retailers. (09/10)

Ultrasonic Cleaner, August 2010: The regulator output voltage at pins 1 & 8 of IC1's socket (ie, with IC1 out of circuit) should be measured with fuse F1 out of circuit. Without the fuse, Mosfets Q1 & Q2 are prevented from providing a partial conduction path that may otherwise blow the fuse or cause some current flow through the transformer. The fuse and IC1 can be inserted with power off once the regulator voltage has been checked (it should be between 4.85V and 5.15V). (12/10)

Hearing Loop Receiver, September 2010: There are some discrepancies between the circuit and the PC board. On the overlay (Fig.2) op amp IC1 is incorrectly shown as an LM358 instead of TL072, as on the circuit. Also, the 10uF capacitor to the left of D2 on the Fig.2 overlay should be reversed in polarity. On the circuit, the anode of diode D2 on the circuit diagram should connect to the anode of ZD1 rather than the 27k resistor. Also on the circuit, the wiper of VR1 should connect to the 1k resistor end rather than the 10uF capacitor. (10/10)

High-Performance Microphone Preamplifier, September 2010: CON3 on the circuit diagram (Fig.3) is shown with its tip and ring connections swapped. (11/10)

Two TOSLINK-S/PDIF Audio Converters, October 2010: The 300-ohm resistor on the S/PDIF to TOSLINK board should be changed to 82-ohm for improved coaxial cable termination. This affects Fig.1, Fig.3, Table 1 and the parts list. (12/11)

Digital Lighting Controller, October-December 2010: (1) The following parts should be added to the master module parts list: 1 28-pin DIP socket, 2 M3 x 6mm machine screws, 2 M3 shakeproof washers, 2 M3 nuts, 1 mini TO-220 heatsink (Jaycar HH8502, Altronics H0630). (11/10)

(2) There are several errors in the PC board overlay diagram on page 28 of the November 2010 issue. The resistor immediately to the left of D4 should be 390 ohms , not 100 ohms. Of the three nearby 10k resistors, the right-most two should be 3k. The circuit diagram in Pt.1 and the boards in kits and from RCS Radio have the correct values. (02/11)

(3) In the Master Unit circuit (pages 40-41 of the October issue), the connections for pins 4 & 5 of IC1 are reversed. The PCB pattern is correct. (06/11)

(4) When operating the unit with four slaves, it may be necessary to leave the 10kΩ "chain length sense" resistor off one of the four slave boards for correct operation. You can check this by plugging in three slaves, then four; if it works with three but not four, remove that resistor from the fourth slave. (02/20)

DAB+/FM Stereo Tuner, October-December 2010: (1) The collector and emitter labels for PNP transistor Q2 are shown transposed on the circuit diagram (Fig.1). In addition, the 100nF capacitors connected to pins 3 & 5 of IC4 should both be 220nF. The parts layout diagram (Fig.3) in Pt.2 is correct. (11/10)

(2) The LM317T regulator on the main circuit (Fig.1, page 27) should be designated REG4, while the LM3940 immediately above it should be designated as REG5. The parts layout diagram in Pt.2 (Fig.3, page 71, November 2010) is correct. (12/10)

(3) In November 2010, on page 77, under preliminary checks, step 3 has REG4 and REG5 mixed up. The LM317T is REG4 while the LM3940 regulators are REG3 and REG5. (05/11)

(4) Notes/Errata published in the May 2013 issue have been removed because they were incorrect.

Wien Bridge Oscillator, Circuit Notebook, December 2010: The RC components wired to S1a are shown to be connected to +12V when in fact they should be connected to the left-hand side of the lamp at half the supply voltage. (01/11)

USB Data Logger, December 2010-February 2011: (1) In order to support baud rates greater than 57.6kbps (ie, up to 0.5Mbps), change the two 100nF monolithic capacitors on the A0/D4 and A1/D5 inputs to 10nF. (01/11)

(2) To improve filtering of the +3.3V supply rail, the 22uF tantalum capacitor on the output of REG1 should be changed to a 220uF 10V low-ESR electrolytic (Jaycar RE-6300). This applies to both the circuit diagram in Pt.1 (December 2010, page 38) and to the overlay diagram in Pt.2 (January 2011, page 35). Note that there are two 22uF tantalum capacitors shown adjacent to REG1 on the overlay. The capacitor on the left is the one to change. The parts list in Pt.1 should be amended accordingly. (02/11)

(3) The specified inductor has been found to be not suitable for the frequencies at which the regulator operates. This can cause excessive current to be drawn at start-up and with low battery voltages. As a result, it should be changed to a 47uH high frequency ferrite choke (Jaycar LF-1100) which fits in its place. Also, a new version of the firmware is now available (v9.92) which, in combination with the new inductor, reduces the current drawn at start-up as well as fixing some other bugs. (04/11)

(4) Revised software reduces start-up power consumption and squashes a few bugs - see May 2011 issue. (05/11)

(5) The Jaycar PS0024 memory card socket used in this project has been discontinued and is no longer available. There is no equivalent so we have updated the PCBs to suit the Altronics P5722 memory card socket. Note that this socket has a metal shield so if the PCB does not have a solder mask, it will be necessary to place a thin plastic insulator under the socket. We have not tested it but the Amphenol 10100708 socket appears to have the same footprint. (01/12)