In the previous article, we discussed the drawbacks of using a simple resistive voltage divider for testing quartz movements. We saw how the load placed on the voltage divider when the step motor coil fired would cause a momentary but significant drop in voltage. Modern electronic quartz watch test equipment use a more sophisticated power supply circuit that is insensitive to the load from the movement under test. In this next article, I'll cover how to construct a simple power supply using common components that will match the performance of the most expensive dedicated test instruments for less than twenty-five dollars.

Figure 1 shows the completed circuit board. The two large 3-lead components that look like power transistors are actually a pair of sophisticated voltage regulator integrated circuits. The design uses an LM317 voltage regulator integrated circuit (IC) to take the twelve volt direct current from the wall adapter and convert it to a variable output voltage. The LM317 is a simple-looking device, with only three connections to the outside world. Inside, however, is a complicated linear electronic circuit designed to sense and maintain an output voltage determined by a pair of resistors connected across two of the three leads. Using the LM317 drops the number of individual electronic components required from over fifty down to less than ten, greatly simplifying construction.

In a perfect world, a single LM317, a couple of fixed resistors, and one variable resistor would be all we need to build our power supply. The LM317 has one drawback, however: the lowest you can drive the output voltage is about 1.2 volts. For most watch testing applications, you need to be able to turn the voltage down to at least .8 volts, and preferably even lower.

The problem is solved by adding a second voltage regulator IC, an LM7805. Like the LM317, the LM7805 is a three-lead device. Unlike the LM317, the LM7805 is a fixed-voltage regulator, preset to put out exactly five volts. The output voltage is fixed and cannot be changed.

We get the zero-to-three volt range by using the difference in the output voltages from each regulator. The five volts from the LM7805 serves as the ground level for the testing voltage, and is connected to the negative battery terminal of the movement under test. The LM317 is set up to provide a voltage adjustable from five to eight volts. This variable voltage is connected to the positive battery terminal of the movement under test. What the movement sees is the difference between the two voltages. When the LM317 is putting out 6.5 volts, the movement is actually seeing only 1.5 volts (6.5 volts - 5 volts = 1.5 volts). By using the 7805 to shift the ground voltage, we can get the range we require by careful selection of the fixed resistors in the circuit. Figure 2 shows the schematic and parts list.

An added benefit of using both the LM317 and LM7805 is that both IC’s have built-in short circuit protection, so if you accidentally short the test leads together, you won’t blow the power supply. Each regulator is rated to provide at least one amp of current, well above the needs of any watch movement. In normal practice, each regulator would have to be mounted on a heat sink to provide their maximum rated current. In our application, however, the current demands are so low that no heat sink is required, and the regulators are simply soldered to the circuit board and the mounting tab provides what little heat dissipation is required.

In case you’re worried that you might not be able to find either of the regulator IC’s, let me put your mind at ease. Both of these regulators are among the most commonly used devices in the electronics industry,
A plain-vanilla 12-volt direct current wall adapter is used to power the circuit. As a rule of thumb for proper operation, the LM317 needs a source voltage at least two volts higher than its output voltage. Since the highest voltage is eight volts from the LM317, the DC wall adapter must provide no less than ten volts. Twelve-volt adapters are more common, and provide a better margin for the regulators to operate with. The wall adapter should be rated for at least 200 milliamperes of current.

The three capacitors in the circuit provide filtering for the incoming DC power from the wall adapter and to eliminate any noise in the output voltages. C1 is a large value electrolytic capacitor with a voltage rating of at least twice the output from the wall adapter. The listed voltage for most unregulated DC wall adapters is an average value. The actual voltage has a ripple to it that can go several volts higher. For example, the 12-volt DC adapter I use for my supply actually measures around 18.5 volts on my bench meter. Depending on the quality of the adapter you use, C1 may not be needed at all. C1 provides a bit more smoothing of the DC voltage ripple and provides a reserve current source on the circuit board to prevent voltage drops when the movement under test pulses its coil.

C2 and C3 are ceramic disc capacitors with a value anywhere from .01 to .1 microfarad. They are connected across the output of each regulator to ground. Their purpose is to “swallow” any noise glitches in the output voltage by providing a direct
path to ground to any voltage spikes that might appear. Like C1, they may not be required at all, but they provide cheap insurance for correct operation and a clean output voltage.

Light emitting diode D1, called an LED for short, serves as a power-on indicator. The 330-ohm resistor R1 limits the current passing through D1 to a safe level. Some LED assemblies come with the current-dropping resistor already installed. In that case, the assembly is connected directly across the two power traces on the board.

As stated before, the output from the 7805 is fixed at five volts, and cannot be varied. The output from the LM317 is set by the ratio of two resistances. The exact equation is:

\[ V_{out} = 1.25 \left( 1 + \frac{R_a}{R_b} \right) \]

Where \( V_{out} \) is the output voltage, \( R_b \) is resistor R2 in our circuit, and \( R_a \) is the combined resistance of R3 and R4. Since R3 is a variable resistor, we can change the value of \( R_a \), and thereby change the voltage. R4 is a fixed resistor, and sets the limit for how low the voltage can be dropped.

A quick word on component values is needed at this point. Like any mass-manufactured item, resistors have a tolerance to their exact resistance value. For example, most resistors you can buy at a local electronics store have a 5% tolerance, meaning that while the actual resistance may be either 5% higher or 5% lower than the marked value. For a resistor marked 1000 ohms, the actual resistance may be anywhere between 950 ohms to 1050 ohms.

Now, the only critical values in the circuit are resistors R2 and R4. The ratio of these two resistors sets the lower output voltage from regulator U1 to no less than 5.09 volts. If the voltage from U1 was to drop below 5 volts, then the polarity to the movement under test would be reversed. Exactly what would happen to a movement seeing a reversed voltage is a matter of speculation. Face it: we’ve all done it at some time, either installing a cell upside down underneath a cell strap or placing the wrong test probes on the contacts. I’ve never seen a movement damaged by reversing the polarity, but I would rather be safe than sorry. Therefore, R2 and R4 are specified as being precision 1% resistors.

Using the exact values specified for R2 (390 ohms) and R4 (1200 ohms), the lowest output voltage from U1 will be 5.09 volts. The combination of resistor tolerance variations which gives the lowest possible output voltage will be when R2 is at the high end of its tolerance (394 ohms) and R4 is at the low end of its tolerance (1188 ohms), which will set the lowest output voltage to 5.02 volts, which is close, but still safe. If 5% tolerance resistors are used, with R2 at the high end (409 ohms) and R4 at the low end (1140 ohms), the lowest output voltage would be 4.73 volts.

Now, having spent the last three paragraphs telling you why you should use 1% resistors for R2 and R4, I can tell you from experience that in most cases regular 5% tolerance resistors can be used. I’ve built six complete circuit boards while developing this article and the assembly instructions, with all six using 5% resistors drawn from the motley assortment of resistors I’ve accumulated over the years. Not one of the boards came in with U1’s lowest output voltage below 5 volts. So the lesson here is: if you can get 1% resistors, then by all means use them, but if they aren’t available, use regular 5% resistors and see what happens. The odds are with you.

Layout and construction of the circuit is not critical. Those with experience may choose to design their own printed-circuit board, or point-to-point wiring techniques can be used. For those of you with little electronic experience, the following step-by-step instructions are provided. The circuit is built using an experimenter’s etched grid-board available from Radio Shack stores, eliminating the need to produce a custom printed-circuit board. The board has a series of holes with pre-etched copper pads on the underside in various sizes, along with two long copper traces used for distributing power along its length. With a little careful forethought, the entire circuit can be built using both the components themselves and several jumpers between the pads to serve as the interconnections. Placement diagrams and detailed, step-by-step instructions are provided. Checkpoints are included in the instructions to verify the work done up to that point.
Form jumpers J1 and J2 like this:

Form all other jumpers flat like this:
**Step-By-Step Assembly Instructions**

Note: “Install” means to put the component in place on the board and solder the leads to the copper pads. Refer to Figure 3 for the parts placement.

1: Locate and install resistor R1 (330 ohm). Save the cut-off resistor leads for the next step.

2: Form jumpers J1 and J2 using the leads cut from R1. Install J1 and J2 where the two jumpers form loops over the top of the circuit board as shown in the parts placement diagram. These two loops provide an easy place to connect your volt-meter leads for testing in later steps.

3: Solder a length of insulated wire to the +12v In connection on the board. Solder the free end of this wire to the positive side of the DC power jack.

4: Solder a length of insulated wire to the –12v In connection on the board. Solder the free end of this wire to the negative side of the DC power jack.

5: Plug the 12 volt DC adapter into a wall outlet and connect the output line to the DC power jack. Using your volt meter, check for the correct polarity across jumpers J1 and J2. J1 should be positive and J2 should be negative. The voltage reading across J1 and J2 should be between 10 and 20 volts DC. Once you have checked for correct polarity and voltage, disconnect the DC adapter from the power jack.

6: Install electrolytic capacitor C1 (470 uF, 35 volts). Pay close attention to correct polarity. Save the cut leads to make jumpers in later steps.

7: Connect the light-emitting diode D1 to the LED+ and LED- connections on the board, paying attention to correct polarity. Connect the cathode lead to the LED- connection.

8: Reconnect the DC adapter to the power jack. The light-emitting diode D1 should light. If not, reverse the connections to D1 and test again.

9: Locate and install jumpers J3, J4, and J5. These and all further jumpers should be installed flat against the circuit board.

10: Locate and install resistor R4 (1.2 Kohm). Save the clipped leads to make jumpers.

11: Find regulator U1 (LM317T). Look closely at the three leads. About 3/16” from the body of the regulator, the leads become narrower. Using needle-nosed pliers, bend each lead 90 degrees downward at the point where the lead narrows.

12: Install regulator U1. The heat-sink tab should be against the circuit board. Once soldered in place, trim the three leads flush with the copper pads.

13: Locate and install resistor R2 (390 ohm). This resistor installs upright, with the top lead folded over. Note that the body of R2 is installed to the right of R4.

14: Locate and install jumpers J6, J7, and J8 flush against the board.

15: Locate and install jumper J9. Very Important: note that the left side of J9 is soldered to a single copper pad that does not connect to anything else at this point.

16: Locate ceramic disc capacitor C2 (.01-.1 uF). Read and understand the rest of this step before actually soldering C2 in place. Refer to Figure 4 for details,

Slide C2 into place on the circuit board. Check to see that C2’s lower lead is in line with the center lead of U1, and the upper lead of C2 is in line with jumper J8. Solder the upper lead in place first, and trim it close to the board.

Looking at the solder-side of the board, bend the lower lead of C2 over so it touches the free end of jumper J9. Solder the lower lead of C2 to the copper pad where it comes through the board. Then, solder the bent part of the lead to the free end of J9 and trim off any excess lead.

17: Find variable resistor R3 (1 Kohm, linear taper). Solder a length of insulated wire to the center terminal. Looking at the back of R3 with the solder terminals pointing up, solder a second length of insulated wire to the left terminal. See Figure 5 for a better picture.

18: Solder the free ends of the wires from R3 to the points marked R3A and R3B on the circuit board. Either wire can go to either point. Once soldered, turn the shaft of R3 fully clockwise.
19: Reconnect the DC adapter to the power jack. Diode D1 should light.

Connect the negative lead of your voltmeter to jumper J2 and touch the positive voltmeter lead to jumper J9. With R3 fully clockwise, the voltmeter should read between 8 to 8.3 volts. While monitoring the voltage, turn R3 counterclockwise. The voltage should smoothly drop down to within 5 to 5.2 volts.

If no voltage is present at J9, check to see that the lower lead of C2 is soldered both to the pad where it comes through the board and to the left side of J9. If this is correct, but the output voltage is wrong, check the location of all jumpers, resistors, and the placement of regulator U1.

Once the voltage on jumper J9 can be varied from 5 to 8 volts, disconnect the DC adapter and voltmeter from the circuit board.

20: Locate and install ceramic disc capacitor C3 (.01 - .1 uF).

21: Locate and install regulator U2 (7805). Note that U2’s leads DO NOT get bent like U1, and that the heat sink tab goes towards capacitor C1.

22: Locate and install jumper J10 flush against the board.

23: Solder a length of insulated wire to the point marked +Vout on the board. Solder a second length of insulated wire to the point marked –Vout on the board.

24: Reconnect the DC adapter to the power jack. Diode D1 should light.

25: Connect the negative voltmeter lead to jumper J2, and the positive voltmeter lead to the wire connected to –Vout. The voltmeter should read almost exactly 5 volts.

26: Disconnect the voltmeter. Turn R3 fully clockwise. Connect the positive voltmeter lead to the wire from +Vout, and the negative voltmeter lead to the wire from –Vout. With R3 fully clockwise, the voltmeter should read between 3 to 3.2 volts. While monitoring the voltage, turn R3 counterclockwise. The voltage should smoothly drop from around 3 volts to around .2 volts.

This completes construction on the circuit board. In the following article, we’ll tie everything together in a single test fixture.