Four Output Bench Supply

Every workbench needs a power supply — this one provides four different outputs.

Larry Cicchinelli, K3PTO

This project is a four output bench power supply. Three outputs (positive voltage) use identical switching regulator circuits that can be set to be any voltage between 3.3 and 20 V. Each output is independent of the others and is capable of up to 1 A. The fourth output is via a negative regulator capable of about 250 mA. The unit I built has two fixed outputs and two variable outputs. You can also make any combination of them variable within the above range.

The only dependency among the outputs is that they are all driven by a single transformer. One of the features of a switching regulator is that you can essentially trade off between voltage and current. The transformer I used is rated at 25 V and 2 A. As such it is good for 50 W. Assuming that the regulator IC being used has an efficiency of 75%, you will have a total of about 37 W available from all power supply outputs. In practical terms this means you can get more current from the outputs than what the transformer is supplying — as long as you stay within the 37 W and the maximum current per regulator.

The regulator I used for the positive supplies is the 3.3 V version of the LM2575. If you examine its data sheet you will see that the only difference among the models is the internal voltage divider. This allows you to design a power supply with a higher output voltage by simply inserting a resistance between the output and the FEEDBACK pin. I selected the 3.3 V version mainly due to its cost relative to the others. With it I can get any voltage from 3.3 V to 20 V from the circuit. You could also use the "adjustable" version, which will then allow you to select any voltage between 1.23 and 20 V.

The regulator is specified for up to 37 V output. Since I have specified 50 V capacitors, I believe you should be able to get up to about 30 V output. If you want to output a higher voltage than 30 V, I recommend that you use higher voltage capacitors. The transformer I am using is rated at 25 V; however, I have measured the loaded output at closer to 30 V ac, so I could probably get up to 25 V from the regulators. You will also need to use the 200 V range of the digital panel meter (DPM).

There is also a high voltage version of the LM2575 that can provide outputs of up to 57 V. I recommend that you use capacitors rated to at least 100 V if you decide to use that version.

A Little Theory

Switching regulators come in essentially three varieties: buck, boost and buck-boost. The positive regulator in this article is of the buck type — the output voltage is less than the input voltage. The main feature of a switching regulator that differentiates it from a linear regulator is that the switcher oscillates. They generally use a form of pulse width modulation (PWM) in order to regulate the output voltage. The rise and fall times of the oscillator are quite fast and the harmonics can cause interference to communications receivers. This is the reason a spectrum analyzer is one of the pieces of test equipment used to characterize a switching regulator. This is certainly not the case with a linear regulator!

Two of the best tutorials I have found on switching regulators are *Application Note 2031* on the Maxim-IC Web site at **www.maxim-ic.com/appnotes.cfm/an_pk/2031** and at **www.national.com/appinfo/power/**

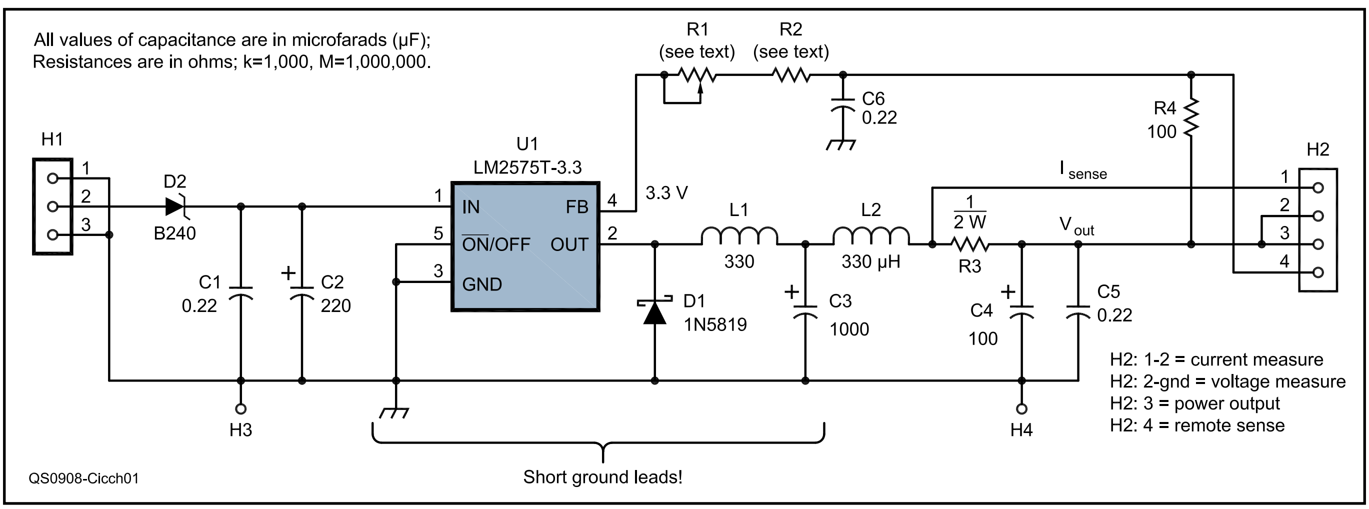

Figure 1 — Schematic diagram of a single positive regulator module.

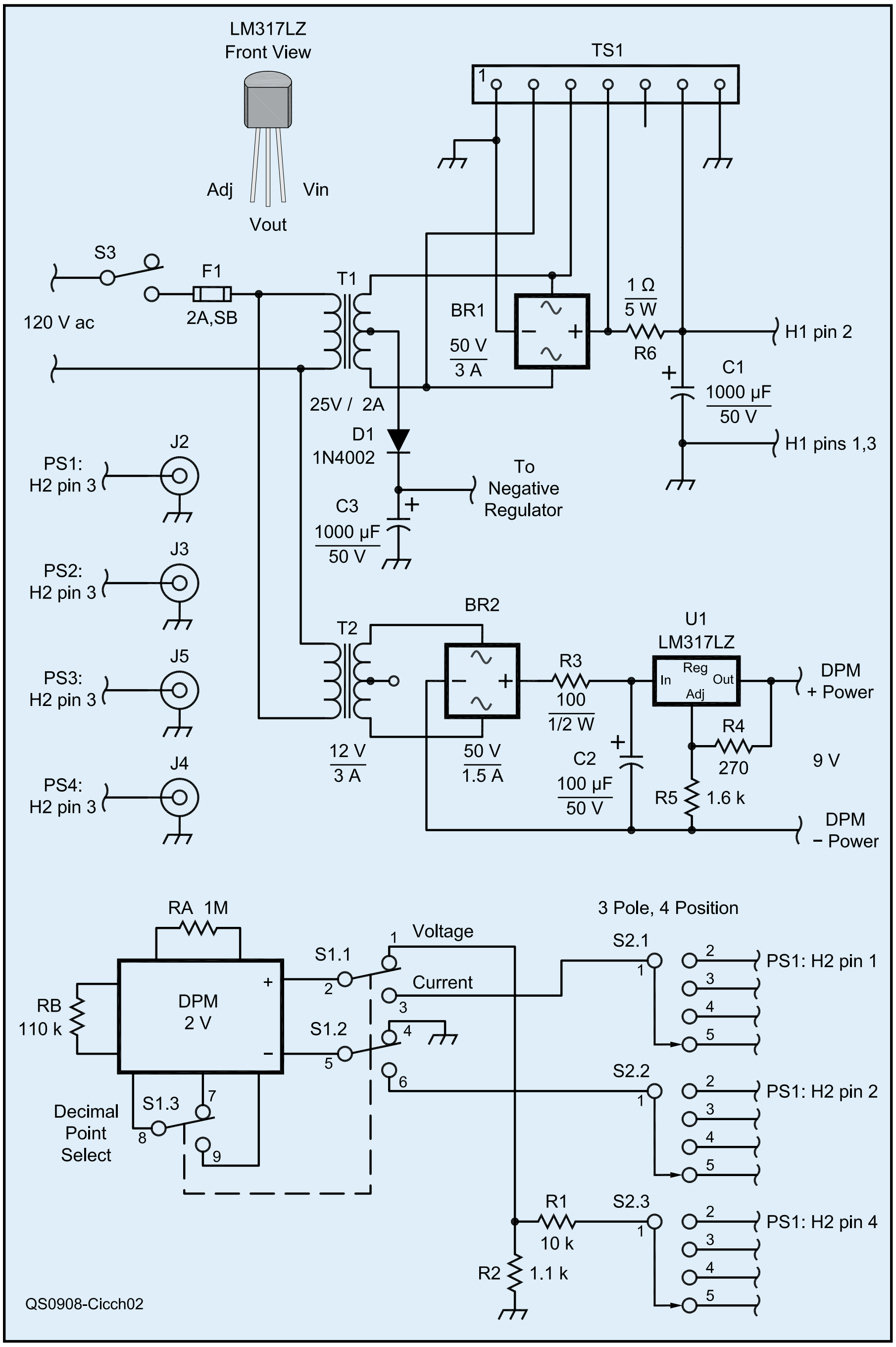

Figure 2 — Chassis schematic showing the interconnection of the modules as well as control and metering details.

files/f5.pdf from National Semiconductor. Rather than try to repeat much of the material in that note, I suggest that you get a copy and read them for yourself.

A switching regulator will have some amount of high frequency noise on its output at the switching frequency, about 52 kHz for the LM2575. In the circuits described here there is a low pass filter on each output that reduces, but does not completely eliminate, this noise. If your requirement is for fixed voltages, you can add a low drop out series regulator (LDO). A good LDO typically requires only about 100 mV between the input and output voltages, so you can design the switcher to be a little higher than the desired voltage and get the benefits of both types of regulators.

Some Circuit Details

Figure 1 is a schematic of a single positive regulator. There are several variations of the circuit which could be implemented. L2 and C4 are optional. These two components provide a low pass filter that will decrease the high frequency noise that might otherwise appear at the output. The pads for R1 will accommodate a small, multi-turn potentiometer. You can insert one here or you can use the pads to connect a panel mounted potentiometer. If you want a fixed output you can simply short out R1 and use R2 by itself. You can also insert a fixed resistor in the R1 position if the calculated value is nonstandard and you want to use two fixed resistors.

The formula for the output voltage (with the 3.3 V version) can be calculated as follows. The current (in mA) through the internal voltage divider is

$$I = 3.3\ \text{V}/2.7\ \text{k}\Omega = 1.22\ \text{mA}$$

$$R1 + R2 = (V_{OUT} - 3.3) / 1.22\ \text{k}\Omega$$

transposing terms yields:

$$V_{OUT} = [1.22 \times (R1+R2)] + 3.3$$

Note that if you make R1=R2=0, the calculation results in an output of 3.3 V. The leakage current of the error amplifier in the regulator is somewhat less than –25 nA, so it can be ignored. Also, since the current for the feedback circuit flows through the current sense resistor, it will be included in the value displayed by the DPM when current is selected.

If you want to have an accurate, fixed output voltage, I recommend selecting a value for R2 that is lower than the calculated value. Then select a potentiometer for R1 that yields a reasonable adjustment range.

If you decide to use the extra LC filter, you will have to install L1 and L2 such that their phasing dots line up with the dot symbols on the circuit board. I found out the hard way that if the dots are at the same end of the board, the output will have an additional low frequency ripple. When I built my board I just happened to have three circuits assembled correctly. The fourth one had a serious low frequency ripple that I could not get rid of. I eventually replaced every component, one at a time, to find the bad one. When I replaced L2 the output was okay. It was then I noticed the phasing dots. I reversed L2 just to see what would happen and the ripple came back. There can be inductive coupling, even though there is a ground plane on both sides of the board under the inductors.

Remote Sensing

A feature of many power supplies is that of remote sensing. This is used to electronically adjust for the voltage drop in the wires carrying current to the load. I found that, even with relatively short wires, there can be significant voltage drop between the regulator and its load. There is provision for remote sensing in this circuit. If you are not going to use remote sensing then you should insert a jumper in place of R4. R4 (100 Ω) is there for protection just in case the remote sense connection is missing.

The schematic shows the connections necessary for remote sensing. It will only work, however, if you run a separate wire from H2-4 to the load. The added accuracy is the result of the load current flowing to the load via H2-3 and essentially no current flowing via H2-4. The connection to H2-2 is still needed in order to measure the voltage drop across the current sense resistor accurately.

If you do not want to use remote sensing you can simplify the switch wiring to use a two pole switch instead of the three pole listed. In this case you would essentially not use S2.2 and connect S1.2 to the common of S2.3 instead of S2.2.

Strictly speaking even this does not fully implement remote sensing. This circuit does not have a mechanism to adjust for the voltage drop in the ground leg. Most high-end commercial supplies will have both power and ground sense inputs. For this power supply make sure that the ground leads have minimal voltage drop. Measurements inside the chassis have indicated this. I have measured about 100 mV drop at 1 A between the positive output of the regulator board and the chassis connector. There was no measurable voltage drop in the ground circuit. You just have to be sure to use relatively heavy wires for the ground connections.

Efficiency

Table 1 on the binaries Web version shows the efficiency of the positive regulator with various input voltages. Notice that the efficiency is really good at 14 V; however, the circuit is no longer regulating! Optimum efficiency seems to occur at 20 V but there is not a whole lot of variation between 16 and 28 V.

Rectifier Circuit

Figure 2 shows the connections among the parts of the system — regulator boards, DPM and rectifier circuit. The components used for the main rectifier circuit are mounted on a terminal strip (Mouser 158-1008). You can see the terminal strip and R6 at the top left of Figure 3. You can hardly see it, but C1 is mounted underneath the terminal strip. The leads of the bridge rectifier are soldered into the holes that are used to rivet the terminals to the Bakelite. One of the four leads, the negative output, is soldered to a grounded terminal. Since I have had quite a few of these terminal strips for several years I used fine Emory paper to clean their surfaces as well as a small file to clean the holes. This was done in order to insure good solder connections.

Negative Regulator

The negative regulator is generally similar to the positive regulator. Its description and schematic are on the binaries Web site.

The Digital Panel Meter

Another feature of the unit is the digital panel meter (DPM). It can be switched to measure the output voltage (H2 pin 1 to ground) as well as the current drawn (voltage between H2 pins 1 and 2) for each of the supplies. Figure 2 shows the circuit I implemented. A three pole, four position rotary switch selects which power supply is being monitored and a three pole, double throw toggle switch selects between voltage and current measurements.

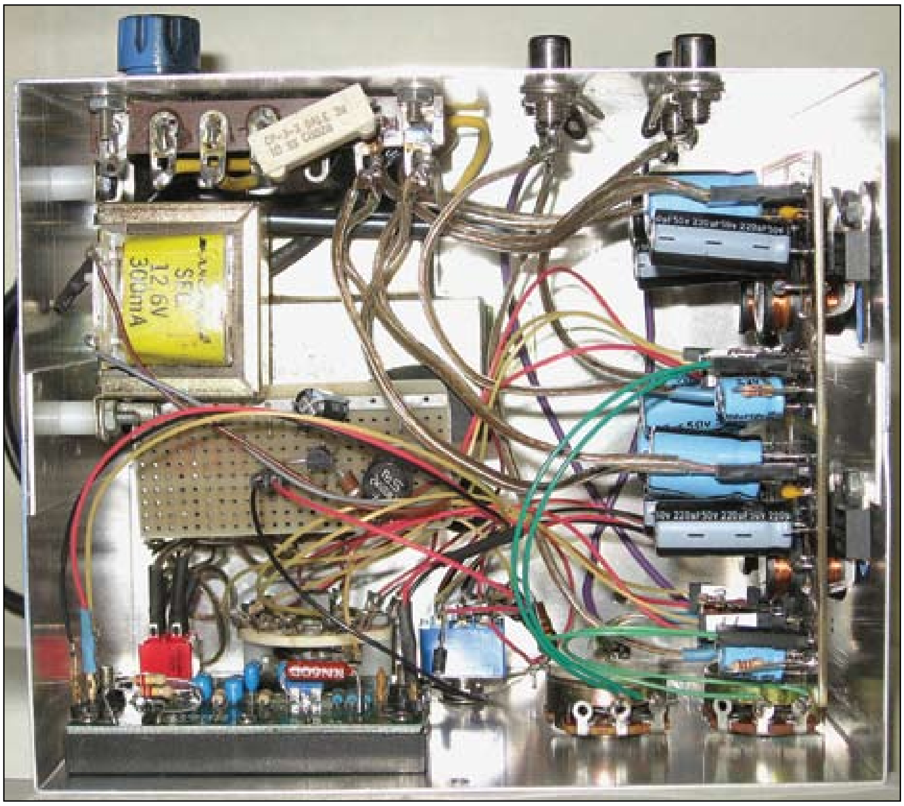

Figure 3 — Underchassis view of the completed power supply. The components used for the main rectifier circuit are mounted on a terminal strip shown at the top left.

The DPM is a 2000 count unit with a basic range of 200 mV. It does not have a 2 V range. I inserted my own resistors on the DPM board for RA (1 MΩ) and RB (111 kΩ). In order to get a 10:1 voltage ratio the resistor ratio needs to be 9:1. If you have to do this for your DPM, you will want to insure that you maintain the accuracy of the meter. I strongly suggest that you maintain at least a 1 MΩ input resistance so that it does not affect the external voltage divider used for measuring the voltage. I used the calibration potentiometer on the DPM for the final accuracy adjustment. I borrowed a four digit DVM of known accuracy to insure good calibration.

It may be hard to get two resistors with exactly a 9:1 ratio from your "junk box." On the DPM I used two 220 kΩ resistors in parallel to get the required 111 kΩ resistance. By measuring several 220 kΩ resistors I was able to find a combination that was quite close to 111 kΩ. For the voltage measuring divider you can do the same thing using a 10 kΩ input resistor and then a 1 kΩ and 110 Ω in series for the "low" side of the divider. The parts list specifies 1% resistors, in case you don't want to combine resistors as I did.

In order to measure the voltage drop across the 1 Ω current sense resistors, the DPM needs either an isolated power supply or some more circuitry (which could require another power supply anyway). For this system I selected the isolated power supply implementation. I used a series regulator because they are somewhat easier to implement and because the DPM has a very low current requirement. Rather than build another printed circuit board I decided to mount all the components, except the transformer, on a breadboard.

The DPM also has a set of jumpers that allow selection of the decimal point location. As can be seen on Figure 2, I use one pole of the toggle switch to select those inputs.

Some Construction Hints

On the DPM and the regulator boards, I used pin headers for all of the connections that come off the boards. (see the parts list for details). This allowed me to assemble the subsystems without having to consider any attached wires. I would then determine the appropriate wire lengths and install the mating connectors on the wires and simply push them onto the pins. This connection method costs about 20 cents per connection.

I have used this method for quite a few

projects and have found it to be very useful. It allows me to disconnect all or part of a circuit for debugging as well as for repair. I typically assemble the connectors under a three power magnifying lens and use a pair of 4 inch needle nose pliers for crimping. A crimping tool would make the job easier but they can be expensive.

I have a supply of eight conductor telephone cable that I use for many of my projects. I cut the cable to an appropriate length and then pull the wires out of the sheath. This gives me wires of eight different colors, making them much easier to trace.

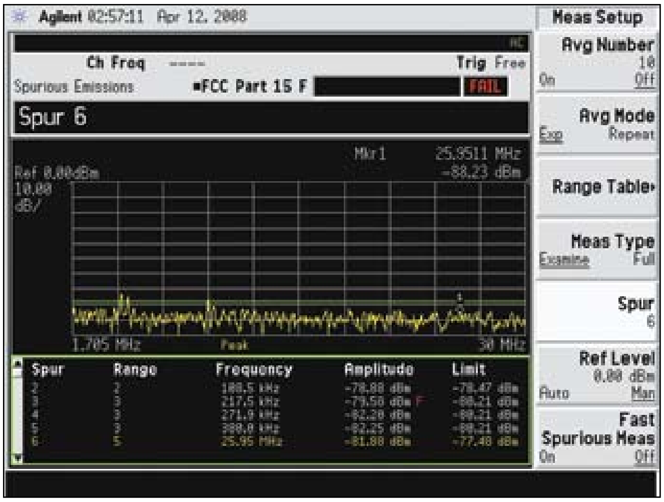

Figure 4 — Measured conducted noise spectrum amplitude throughout the range of 1 to 30 MHz. These show a very quiet output.

The printed circuit board for the regulators contains four identical circuits. At first I thought I might separate the boards for mounting in the enclosure. I decided to keep them as a single board, however. This made it easier to mount the board and it also gave me an idea as to how to heat sink the regulator ICs. I mounted them on the bottom side instead of on the top as I had originally planned. I then folded over the ICs so that the flat side was parallel to the PCB and farthest from the board. I managed to fold the ICs identically so that I was able to use their mounting holes to fasten them to the side of my enclosure. This not only is a convenient method of mounting the board it also gives the ICs a good heatsink and ground. Figure 3 shows the completed assembly bolted to the side of my enclosure.

One problem I encountered with the above method was that of mounting the nuts and bolts required by the ICs. I solved this problem by using a small amount of epoxy to attach the nuts to the "front" side of the regulators. I used a short bolt to hold the nut in place while the epoxy hardened being careful to avoid getting any epoxy on the threads. Figure 5 on the binaries version shows the back side of the PCB assembly with the ICs folded over and the bolts holding the nuts in place while the epoxy hardens.

Another issue I had to solve was the length of the bolts. I had to insure that they were short enough so that they would not touch the PCB once they were used to bolt the ICs to the chassis. Since I did not have bolts of the proper length I had to cut them to length. I used a pair of bolt cutters with threaded holes for the 10-32 bolts. A word of caution here — to ensure that I could still thread a nut onto the bolt after cutting it, I threaded a nut onto each bolt beforehand, so that after I cut it I would have to remove the nut. This method helps to insure that the bolt will still allow a nut to be threaded onto it.

I then measured the distance between the mounting holes of the four regulator ICs and drilled holes in the side of my enclosure accordingly. This method proved relatively easy to implement and made for a very simple mounting procedure. I suggest making a drilling template out of card stock (an index card, for example) to ensure correct spacing, before you drill any holes.

Since the negative regulator was an addition to the "completed" system its installation was different from that of the positive regulators. The circuit was built on a prototyping breadboard and bolted to the bottom of the enclosure. I removed the connections to my regulator #3 and rerouted them to the negative regulator. I have since designed a printed circuit board, but have not replaced the one in my system.

For those who may already have a positive voltage power supply, the negative regulator can be constructed as a separate project and simply connected to your existing supply.

A caution regarding the circuit boards. My source is FAR Circuits, which makes a lot of boards for ham related projects. Their boards do not have plated-through holes so you will have to be sure that you solder the through-hole components on both sides of the board.

The QST Binaries Web site has the artwork for the board as well as the Gerbers and a drill file.[1] These can be used to make your own boards if you want. The schematic capture software I use is *DipTrace*. The schematic and PCB files are also on the binaries site.

RFI

I have access to a really good spectrum analyzer at my employers, as well as someone who knows how to use it (thank you, Matt!). The power supply, in its aluminum enclosure, was put into a completely shielded box with an 18 inch antenna within a few inches of it. I put an 8 Ω, 20 W resistor on the #2 regulator output and adjusted the voltage to +8 V. We then made a series of measurements. The spectrum analyzer has a mode in which it does FCC Part 15 (RFI) tests automatically — very convenient! Even with the antenna so close, the only interference that would have failed the test occurs around 88 MHz). The horizontal green line shows the FCC limit of about –62 dBm. Figure 4 shows conducted noise — the analyzer probe was connected directly to the output through a 0.22 µF capacitor. Figure 4 shows the noise throughout the range of 1 to 30 MHz. This shows a very quiet output.

Parts

The only critical parts are the resistors that form the two voltage dividers. Even their values can be changed, within reason, as long as the ratios are maintained. Although the value of C3 is not very critical, it should be a low ESR type.

The parts list (also on the binaries site) provides the sources from which I obtained my parts. Since I have built quite a few projects over the years I have developed a fairly good supply of components. I have a spreadsheet I keep updated with everything I purchase so that I can use the same parts in new projects. Many of the parts can be obtained from several sources so you may want to do a little shopping around. I try to minimize costs by getting parts from as few sources as possible in order to keep shipping costs down.

Photos by the author.

Larry Cicchinelli, K3PTO, is an ARRL member who has been licensed as K3PTO since 1961. He holds an Advanced class license. Larry earned a BSEE from the Drexel Institute of Technology in 1969 and an MSES from The Pennsylvania State University in 1981. He was employed at Ford Motor Company for 33 years until 2000, responsible for the design and fabrication of test equipment first for ICs and then for automotive electronics. He is now Technical Support Manager for Rabbit Brand at Digi International. He has had articles published in QEX, Circuit Cellar *and* Nuts & Volts *magazines. You can reach Larry at 119 River Run Cir, Sacramento, CA 95833 or at* **k3pto@arrl.net**.

[1]www.arrl.org/files/qst-binaries/