This project describes a four output bench power supply. Three outputs (positive voltage) use identical switching regulator circuits which you can set to be any voltage between 3.3V and 20V. Each output is independent of the others and is capable of up to 1 amp. The fourth output is a negative regulator and is capable of about 250ma. The unit I built has two fixed outputs and two variable outputs. You can also make any combination of them variable within the 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 25V and 2 amps. As such it is good for 50 watts. Assuming that the regulator IC being used has a 75% efficiency, that means that you will have a total of about 37 watts available from the power supply outputs. In practical terms this means that you can get more current from the outputs than what the transformer is supplying – as long as you stay within the 37 watts and the maximum current per regulator. Most of the discussion in this article will be about the positive regulators as the negative regulator came as an add-on after the original system was built.

The regulator I used for the positive supplies is the 3.3V 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 3.3V version mainly due to its cost relative to the others. With it I can get any voltage from 3.3V to 20V from the circuit. If you want, you can use the “Adjustable” version which will then allow you to get any voltage between 1.23V and 20V.

The regulator is specified up to 37 volts output. Since I have specified 50V capacitors I believe you should be able to get up to about 30V output. If you want to go higher than 30V I recommend that you use higher voltage capacitors. The transformer I am using is rated at 25V; however, I have measured the output at closer to 30VAC (when loaded) so I could probably get up to 25V from the regulators. You will also need to use the 200V range of the DPM (unless you like to multiply readings by some factor).

There is also a high voltage version of the LM2575 which will allow you to achieve outputs up to 57V. I recommend that you use capacitors rated to at least 100V if you decide you want to do this.

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 in 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 website: http://www.maxim-ic.com/appnotes.cfm/an_pk/2031 and http://www.national.com/appinfo/power/files/f5.pdf by 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 52KHz for the LM2575. In the circuits described here there is a low pass filter on each output which reduces this noise but it does not eliminate it. If your requirement is for fixed voltages you can add a Low Drop Out series regulator. A good LDO typically requires only about 100mv 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

Schematic-1 is the circuit for one positive regulator. There are several variations of the circuit which you can implement.

- L2 and C4 are optional. These two components implement a low pass filter that will decrease the high frequency noise which 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 in the case where the calculated value is non-standard and you want to use two fixed resistors.

The formula for the output voltage (with the 3.3V version) can be calculated as follows:

current (in ma) through the internal voltage divider = 3.3V/2.7K = 1.22ma
R1 + R2 = (Vout – 3.3) / 1.22 (resistance in Kohms)
transposing terms yields: Vout = 1.22(R1+R2) + 3.3

Note that if you make R1=R2=0, the calculation results in an output of 3.3V. The leakage current of the Error Amplifier in the regulator is somewhat less then -25na 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 that you select a value for R2 which is lower than the calculated value. Then select a potentiometer for R1 which yields a reasonable adjustment range.

If you decide to use the extra LC filter, you will have to install L1 and L2 such 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 OK. It was then I noticed the phasing dots. I reversed L2 just to see what would happen and the ripple came back. So, there can be coupling even though there is a ground plane on both sides of the board under the inductors.

Remote Sensing

Another 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 relative 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 (100Ω) is there for protection just in case the remote sense connection is missing.

To do remote sensing most effectively you will have to implement the circuit somewhat differently than is indicated in the chassis schematic. I used the same point to pick up the output voltage for both the voltage and current measurements (H2-2). This measures the voltage at the output of the regulator board – not at the load. If you want really accurate readings on the DPM and have accurate remote sensing you will have to pick up the voltage measurement from the remote sense input of the board at H2-4 (see Schematic-3 Chassis). This will involve using a 3 pole 4 position rotary switch. This is necessary because you still need to measure the voltage drop across the current sense resistor right at the resistor.

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 Sw2.2 and connect Sw1.2 to the common of Sw2.3 instead of Sw2.2.

Strictly speaking even this does not implement full 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 I am assuming that the ground leads have minimal voltage drop. Measurements inside the chassis have proven this. I have measured about 100mv drop at 1 amp 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

This chart shows the efficiency of the positive regulator with various input voltages. Notice that the efficiency is really good at 14V; however, the circuit is no longer regulating! Optimum efficiency seems to occur at 20V but there is not a whole lot of variation from 16V to 28V.

Positive Regulator: Rload = 50 ohms

Measured Values			Calculated Values		
Vin	Iin	Vout	PwrIn	PwrOut	Efficiency
14	0.275	13.7	3.85	3.75	98%
16	0.25	12.6	4	3.18	79%
18	0.23	12.6	4.14	3.18	77%
20	0.2	12.6	4	3.18	79%
22	0.19	12.6	4.18	3.18	76%
24	0.175	12.6	4.2	3.18	76%

26	0.165	12.6	4.29	3.18	74%
28	0.155	12.6	4.34	3.18	73%

Rectifier Circuit
Schematic-3 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 Picture-2. You can hardly see it, but C1 is mounted underneath the terminal strip. The leads of the bridge rectifier are soldered into the holes where the terminals are riveted 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 of the buck-boost configuration – it converts a positive voltage into a negative one – see Schematic 2. This design uses many of the same component values as the positive regulators. For this circuit I had to switch to using an LM2673 because I could not stabilize the output voltage using the 2575. It could have been my circuit layout but the 2673 worked well roght off the bat! I was unable to implement the current measuring circuit within a feedback loop. I tried several configurations but each introduced a significant low frequency noise component to the output voltage. There was also some 250 KHz noise present on the output so I implemented an additional low pass filter on the output which reduces it considerably.

The negative regulator uses a different rectifier than the positive regulators. This was necessary because the LM2673 has a maximum input to output voltage specification of 40 volts. Since the output of the main bridge circuit is 30V and the output of the negative regulator is -15V, the difference exceeds the specification. Using the center tap of the transformer yields close to +16V for the negative regulator. The half wave rectifier circuit is not as efficient as a bridge but it will suffice for benchtop analog circuits. The components can be mounted to the unused terminals of the terminal strip used for the main rectifier.

One surprising feature of the circuit is that there is considerable inductor current even with no load. The formula for the equivalent circuit using a LM2575 is on page 20 of its data sheet. One of the cited references indicates that this is typical of this type of circuit – the input voltage is applied across the inductor for 50% of the time.

Even though I designed this circuit for a load of up to about 250ma I had to use a value of 22K for the current limiting resistor. This should limit the current to about 2amps. The original value I used of 39K, a limit of about 1amp, should have been fine. However, I found that the circuit would not regulate properly with the 39K. I really do not know why!
Here is chart showing the efficiency of the negative regulator with various input voltages. As you can easily see the efficiency is better with an input voltage of at least 10 volts.

Negative Regulator: Rload = 50 ohms

Measured Values			Calculated Values		
Vin	Iin	Vout	Pin	Pout	Efficiency
8	0.466	11.3	3.728	2.55	69%
10	0.573	14.78	5.73	4.37	76%
12	0.475	14.78	5.7	4.37	77%
14	0.409	14.78	5.726	4.37	76%
16	0.356	14.78	5.696	4.37	77%

The Digital Panel Meter

Another feature of the unit is the DPM which can be switched to measure the output voltage (H2 pin 1 to ground) as well as the current draw (voltage between H2 pins 1 and 2) for each of the positive supplies. Schematic-3 shows the circuit I implemented. A 3-pole, 4 position rotary switch selects which power supply to monitor and a 3PDT toggle switch selects between voltage and current.

The DPM is a 2000 count unit with a basic range of 200mv and does not have a 2V range. I inserted my own resistors on the DPM board for RA (1M) and RB (111K). 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 1M 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 good, 4 digit, DVM 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 220K resistors in parallel to get the 111K resistance. By measuring several 220K resistors I was able to find a combination which was quite close to 111K. For the voltage measuring divider you can do the same thing: use a 10K input resistor and then a 1K and 110 ohms in series for the "low" side of the divider. The Bill of Materials lists 1% resistors so that you do not need to combine resistors as I did.

In order to measure the voltage drop across the 1 ohm 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 simply because they are somewhat easier to implement and the very low current requirement of the DPM. 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 you to set the decimal point location. As can be seen on the Schematic-3 I use one pole of the toggle switch to select its location.

Some Construction Hints

On both the DPM and the regulators I used pin headers for all of the connections which come off the boards (see the BOM 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¢ 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 3X magnifying lens and use a pair of 4" 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 eight different color wires making it much easier to trace them once they have been installed.

The printed circuit board for the positive regulators contains four identical circuits. At first I thought I might separate the boards for mounting in the enclosure. However, I decided to keep them as a single board. 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! Picture-2 shows the completed assembly bolted to the side of my enclosure.

One problem I encountered with the above method was that of "How do I mount 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. Picture-4 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 insure that I could still thread a nut onto the bolt after cutting it, I threaded a nut onto each bolt before hand, 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. You might want to make a drilling template out of card stock (an index card) to ensure correct spacing before you drill any holes.

Since the negative regulator was an addition to the "completed" system its installation was different as compared to 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 re-routed them to the negative regulator. I have since designed a printed circuit board but have not replaced the one in my system.

For those of you who may already have a positive voltage power supply you can build the negative regulator circuit and simply connect its input to the output of your existing supply.

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

A zip file available on my web site (www.qsl.net/K3PTO) has the artwork for the board as well as the Gerbers and a drill file. These can be used to make your own boards if you want. The schematic capture software I use is DipTrace©. If you would like the schematic and PCB files please email me and I will make them available.

RFI

I have access to a really good spectrum analyzer where I work, 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" antenna within a few inches of it. I put an 8Ω, 20W resistor on the #2 output and adjusted the voltage to +8V. We then made a series of measurements. The spectrum analyzer has a mode where it does FCC Part 15 tests automatically – very convenient! Even with the antenna so close the only interference that would have failed the test occurs around 88MHz. (See Picture 5.) The horizontal green line shows the FCC limit of about -62dbm. Pictures 6 and 7 show conducted noise – the analyzer probe was connected directly to the output through a .22uf capacitor. Picture 6 shows the amplitude around the oscillator frequency and Picture 7 shows the noise throughout the range of 1MHz to 30MHz. Even these show a very quiet output.

Parts

The only critical parts are the resistors which form the two voltage dividers. Even their values can be changed, within reason, as long as the ratios are maintained. The value of C3 on the circuit is not real critical; however, it should be a low ESR type. I kept the circuit designations for both regulators the same for the equivalent parts. This makes it easier to reference the components of the two circuits.

The BOM lists the sources where 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. Sometimes it is cheaper to buy a part from a vendor at a slightly higher price just to save on shipping costs.