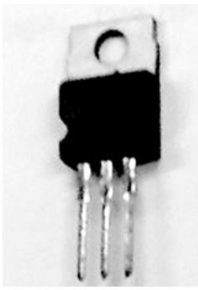
Adjustable Linear Regulators

As nice as it is to be able to buy a simple device that produces a constant output voltage, the Fixed Linear Regulators are limited to the output voltages selected by the manufacturer. The Adjustable Linear Regulators provide the possibility to produce other voltages, and, using potentiometers in the feedback, to provide a range of output voltages from the same device. Benchtop power supplies use adjustable regulators to allow the user to set the output voltage to anything they want, within the limits of the equipment.

The most common Adjustable Linear Regulators are the positive in - positive out LM317 and the negative in - negative out LM337.

These two devices have the same pinout, which is nice, but the pinout is not intuitive, so many students wire these up incorrectly and burn them out. Be careful!

Again, these typically come in a TO220 Package.



TO220 Package

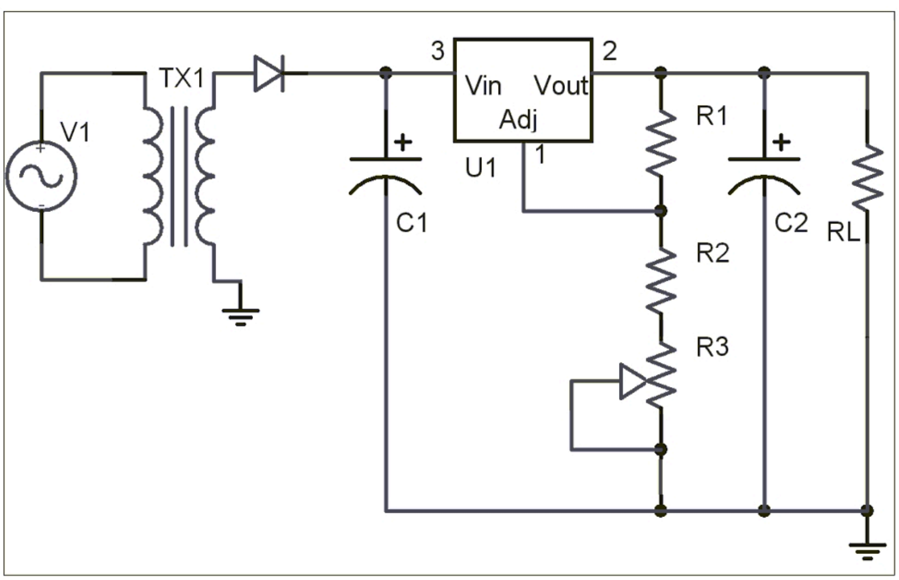
Pin	Function
1	Adj
2	Vout
3	Vin
Metal Back	Vout

Three things are important when using these devices in a design:

1. The dropout voltage is 2.5 V
2. The voltage between Vout and Adj is designed to be 1.25 V
3. There is a small, constant leakage current of about 50 μA, from the Adj pin

Point #2 is the most important one -- it's the one we use to come up with a workable design. However, the other two points need to be taken into account in order to avoid nasty surprises.

Here's a typical design using an adjustable linear regulator.



Notice that, since the regulator is not directly referenced to ground, the leakage current from pin 1, Adj, passes through R2 and R3, and will therefore produce an error voltage in the design. Let's investigate:

1. If the combined value of R2 and R3 is 2.0 kΩ, what error in the output voltage would be introduced by the 50 μA leakage current? 0.1 V
2. If, however, the combined value of R2 and R3 is 10 kΩ, what error in the output voltage would be introduced? 0.5 V

Clearly, to minimize the error introduced by the leakage current, the resistance between the Adj pin and ground must be kept small. However, the 1.25 V between the Vout and the Adj is usually smaller than the voltage across R2 and R3, so that means R1 is typically smaller, and sometimes significantly smaller, than the R2, R3 combination. As a rule of thumb, we aim for a combined resistance of R2 and R3 of 2 kΩ or slightly less to give us some room to pick suitable resistors.

Question: For the circuit above, assume V1 is 120 VAC at 60 Hz, the transformer turns ratio is 10:1, the rectifier diode has a forward voltage drop of 0.7 V, C1 is 680 μF, C2 is 10 μF, and the minimum resistance of RL is 100 Ω.

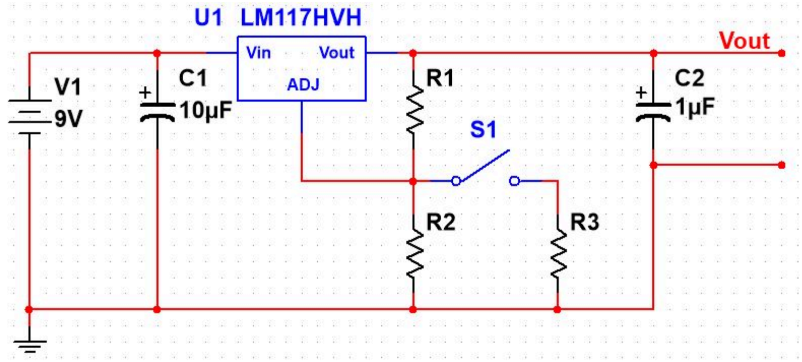
1. Which adjustable regulator should we use for U1? LM317
2. What is the peak voltage at the output from the rectifier? 16.3 V_p
3. Starting with a combined value for R2 and R3 of 2 kΩ, calculate a value for R1 that will produce an output voltage of 8,192 mV. 360 Ω
4. Locate a list of 10% resistor values (there's one on the formula sheet for this course). Since there isn't a 10% resistor value that exactly matches the calculated value, pick the next smallest one (picking a bigger one would mean we would have to increase the R2, R3 combination, which we don't want to do) 330 Ω
5. Now for the R2, R3 combination. Typically, to allow for fine tuning, we make the fixed resistor 2x to 5x bigger than the potentiometer. Potentiometers, being adjustable, don't come in a lot of values -- typically multiples of 1, 2, or 5. Usually, we would play with the numbers until a suitable set is arrived at, but to help with Mobius grading, let's start with the assumption that the fixed resistor will be 3x the size of the potentiometer. Fixed resistor = 1.5 kΩ, Potentiometer = 500 Ω
6. In reality, we would simply adjust the potentiometer until the output is what we want it to be. What would the combined resistance of R2 and R3 be, approximately, when the output is adjusted to 8192 mV? 1.83 kΩ
7. What is the maximum load current, once the voltage is set to 8192 mV? 81.9 mA
8. What ripple voltage would be seen at the output from the rectifier diode? 2.01 V_{p-p}
9. Using this ripple voltage, what voltage would the unregulated input signal drop to each cycle? 14.3 V
10. Is this likely to cause dropouts in the output signal? No

Worked Example: Selectable 3.3 V / 5 V Regulator

For this circuit, a client has asked for a regulator operating from a 9 V battery that has a switch to select between 3.3 VDC and 5 VDC. Since the LM317 requires an overhead of 2.5 V, we'll have no trouble using a 9 V battery, because it will give us an overhead of 4.0 V when generating 5 VDC, which is ample.

Assuming that the choice of voltages means that this will be a power supply for TTL-level devices (5 V) or LVTTL-level devices (3.3 V), there will be a range of allowable voltages: "5 V" for TTL can range from 4.5 V to 5.5 V, and "3.3 V" for LVTTL can range from 2.7 V to 3.6 V. We'll aim for values equal to or slightly higher than the nominal voltages.

For this design, we'll pick a fixed value for the resistor between the Vout and ADJ pins of an LM317-style regulator, and we'll provide two different values for the other resistance by switching in a second resistor in parallel, as shown in the following schematic. The LM117HVH is a military-specification version of the LM317, and is used in this schematic because it's the only version of the IC that's available in Multisim.



The larger the resistance between ADJ and ground, the higher the output voltage. So, we'll start our design for 5 VDC, as that will only involve R1 and R2 in the calculations. That way, we can make R2 a reasonable size but still less than the 2 kΩ maximum needed to minimize the efffect of leakage current.

The voltage divider formula used for the feedback network can be rearranged to provide us with the ratio of the resistor values:

$$V_{reg} = 1.25\left(\frac{R_1+R_2}{R_1}\right)$$

rearranges to

$$\frac{R_2}{R_1} = \frac{V_{reg}-1.25}{1.25}$$

For 5 V, then, R_2/R_1= 3.00

Playing around with the various 10% resistor values in the kit, the closest we can get to this ratio is R_2 = 1.2 kΩ and R_1 = 390 Ω. To see how close our theoretical results (based on the chosen values) are to the ideal value, we'll enter the resistors into the original formula. When we do this, we get V_{reg} = 5.1 VDC, which meets our specifications.

For 3.3 V, we'll need to use the same R_1 -- 390 Ω -- and determine what resistor we'll need to have in parallel with R_2 to get close to 3.3 V.

The ratio of resistances gives us, for 3.3 V, R_2/R_1 = 1.64

So that means the parallel combination of R_2 and R_3 needs to be 1.64*390 Ω, or 640 Ω

Rearranging the parallel resistor calculation is a bit of a challenge -- give it a try!

$$R_T = \frac{1}{\frac{1}{R_2}+\frac{1}{R_3}}$$

rearranges to

$$R_3 = \frac{R_2 \cdot R_T}{R_2 - R_T}$$

... which produces a value of 1.37 kΩ for R_3 using our chosen value of 1.2 kΩ for R_2. 1.37 kΩ is halfway between two 10% resistor values, so we'll choose the larger value to bump the voltage up slightly. We'll pick R_3 = 1.5 kΩ.

Again, we'll check to see that the theoretical output for this resistor is to see if it matches the specification. We get 3.4 VDC, which is suitable.