

CK402 - ADJUSTABLE VOLTAGE POWER SUPPLY

This is a basic, text-book, adjustable voltage regulator circuit using an LM317, 3-terminal regulator in a TO-220 package. The output voltage can be set to anywhere in the range 1.5V to 30V by selecting two resistances. By using a potentiometer, R2, as one of the resistors you can dial up the output voltage wanted. Either AC or DC input can be supplied to the Kit via a socket or terminal block. This is because we have provided a bridge rectifier on board. The input DC voltage to the regulator must be at least 2.5V above the required output voltage. An off/on switch is provided. For many applications (say 12V at 60mA) a heat sink will not be necessary. However, one has been provided in the kit.

The LM317 will provide slightly higher output voltages than 30 volts. However, for most hobbyists over 30V will not be needed. So to make a small PCB we have used some electrolytic capacitors rated to 35 volts. To be safe for continuous operation the maximum input DC voltage to the regulator should not be over 33V. With a 2.5V to 3.0V drop across the regulator this will give a regulated output of 30V. You can draw up to 1.5A from the LM317. If you need higher then use an LM338T rated to 5A.

When external capacitors are used with any IC regulator it is good practice to add protection diodes to prevent the capacitors discharging back into the regulator in the event of abnormal operating conditions, like a sudden short circuit on the input or the output, or a back emf from an inductive load. That is the function of D1 and D2.

The value of R1 can range anywhere from 120R to 1200R. However, circuits from most other sources settle on using either 220R or 250R. We have used 240R or 250R. The voltage drop across R1 is 1.25V for all values, and this is the key to the design. 1.25V is the reference voltage of the regulator. Whatever current flows through R1 also flows through R2, and the sum of the voltage drops across R1 and R2 is the output voltage. (Additional current I_d also flows in R2 but it is typically 50uA so is negligible.) The design formula are:

$$V_{OUT} = 1.25 (1 + R2/R1) \text{ volts, or alternatively}$$

$$R2/R1 = (V_{OUT}/1.25) - 1$$

So if you know V_{OUT} and R1 is 250R then you can calculate R2. If you find that the 5K potentiometer used for R2 does not give you the degree of fine control over the voltage output range that you want then you can use these formulae to adjust R1 and R2 to better suited values.

Construction. Check all the components supplied against the Components listing above. It is usually best to solder the lowest height components onto the PCB first and then do to the taller components. Note the polarity of the diodes and capacitors. The back of the LM317T is marked on the overlay.

COMPONENTS		
2 pole edge connector		2
1N4004 diode	D1 D2	2
Mono cap .1uf (104)	C2	1
5K potentiometer	R2	1
Electrolytic capacitors:		
1000uF/35V	C1	1
10uF/50V	C3	1
100uF/35V	C4	1
Resistor 240R or 250R, 5%	R1	1
LM317T regulator		1
Bridge rectifier W02		1
SPDT switch		1
Kit 68 PCB		1
PCB mounted power jack		1
Heatsink		1
Nut & bolt set 6mm or 8mm		1

For other kits go or to contact us go to our website at <http://www.electronickits.com>

