TWO INDIVIDUAL VARIABLE DC POWER SUPPLY

PRODUCT CODE: M00270038

DESCRIPTION: Two individual variable DC power channel is provided by a single channel DC power supply.

READ BEFORE INSTALLATION:

- Put the component on the side of screen printing and solder on the back of PCB without printing.
- Placing direction of component.

1. On component, longer leg is "+".
2. On PCB marking, square pad as Figure 1 is always "+".
3. For diode, please install as Figure 2.

- Do not put the LED to very bottom, just install as Figure 3.
- For any IC, finding out which leg is first leg (FIGURE 4) is important. Also, solder the socket (chair) to the PCB and the IC sit on the top.
- For 9V Battery Adaptor, Red is B+ and Black is B-. Also, please tie a knot after the red and black wire has passed the neighbors hole before soldering. This is similar to Figure 6.

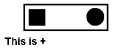

FIGURE 1

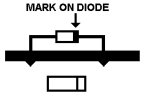

FIGURE 2

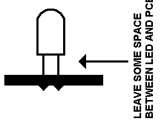

FIGURE 3

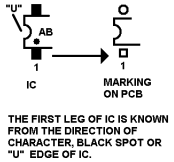

FIGURE 4

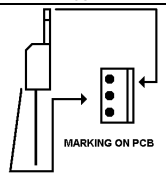

FIGURE 5

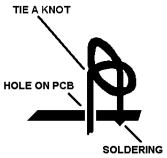

FIGURE 6

FIGURE 7

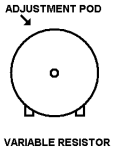

FIGURE 8

FIGURE 9

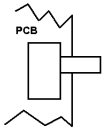

FIGURE 10

FIGURE 11

CIRCUIT EXPLANATION:
Please read the below together with the circuit diagram in Figure 12.

Part 2 is the major body of this circuit. I assume the voltage of leg 3 of U1A is stable due to part 1. One of the functions of V2 is the voltage sensing part of the output voltage. At any moment, I assume the output voltage is little lower. Then this made the feedback voltage to leg 2 of U1A becoming little lower. As a result, the voltage different between leg 3 and leg 2 would be bigger. Because U1A is an amplifier, finally leg 1 would be little higher. The result is that Q3 would open more, more current would flow to Q1 and made Q1 also open more. Then, more current would flow to output. The output voltage would become little higher and go back to normal. From the circuit, you can see the second function of V2 is the voltage control at the output because the sensed voltage at the output can be adjusted by this variable resistor.
What would happen when output voltage is little higher? In this case, Q3 and Q1 would be close due the logic on above. The voltage would go down due to the loss of charge from C2 and C1 to the load. Until the feedback voltage from V2 to leg 2 is little lower than leg 3. Q3 and Q1 are open again.
We now go back to part 1, this is to made the reference voltage of leg 3 of U1A always stable. This is done by using the zener diode D1. Then why we do not connect the zener diode directly to leg 3 of U1A? The reason is that the lowest voltage reference of zener diode we can buy is around only 2V. If you want the output voltage is lower than 2V, the voltage at leg 3 must be less than 2V. By using V1, we can adjust the voltage at leg 3 lower than 2V.
The function of C1 at Part 3 is to make the output voltage more stable. C2 is to filter the AC voltage from the load.
Part 4 is the second channel. This is the same as Part 1 to Part 3.

Part 5 is for input of DC voltage (NOTE 1). D3 is to prevent the wrong connecting of polarity of DC power supply.

INSTALLATION:
Just install the component to the PCB M00260063 according to below table.

ITEM	SYMBOL ON PCB	DESCRIPTION	OUTLOOK	DIRECTION IS IMPORTANT?
1	R1	RESISTOR, 10K ohms	BROWN, BLACK, ORANGE	NO
2	R2	RESISTOR, 100K ohms	BROWN, BLACK, YELLOW	NO
3	R3	RESISTOR, 10K ohms	BROWN, BLACK, ORANGE	NO
4	R4	RESISTOR, 100K ohms	BROWN, BLACK, YELLOW	NO
5	R5	RESISTOR, 330 ohms	ORANGE, ORANGE BROWN	NO
6	L1	LED	ONE LONG LEG AND ONE SHORT LEG	YES
7	D1	ZENER DIODE, 2.2V	FIGURE 2 (MOSTLY TRANSPARAENT RED)	FIGURE 2
8	D2	ZENER DIODE, 2.2V	FIGURE 2 (MOSTLY TRANSPARAENT RED)	FIGURE 2
9	D3	DIODE, IN4001	FIGURE 2 (MOSTLY BLACK)	FIGURE 2
10	Q1	TRANSISTOR, NPN	FIGURE 5, MARK WITH TIP41	YES
11	Q2	TRANSISTOR, NPN	FIGURE 5, MARK WITH TIP41	YES
12	Q3	TRANSISTOR, NPN	FIGURE 9, MARK WITH 9014	YES
13	Q4	TRANSISTOR, NPN	FIGURE 9, MARK WITH 9014	YES
14	Q5	TRANSISTOR, NPN	FIGURE 9, MARK WITH 9014	YES
15	Q6	TRANSISTOR, NPN	FIGURE 9, MARK WITH 9014	YES
16	V1	VARIABLE RESISTOR, 10K ohms	FIGURE 11	YES
17	V2	VARIABLE RESISTOR, 10K ohms	FIGURE 8	YES
18	V3	VARIABLE RESISTOR, 10K ohms	FIGURE 11	YES
19	V4	VARIABLE RESISTOR, 10K ohms	FIGURE 8	YES
20	C1	CAPACITOR, 1000uF	MARK WITH 1000uF OR SAME MEANING OF VALUE	YES
21	C2	CAPACITOR, 22*10E4 pF	MARK WITH 224 OR SAME MEANING OF VALUE	NO
22	C3	CAPACITOR, 22*10E4 pF	MARK WITH 224 OR SAME MEANING OF VALUE	NO
23	C4	CAPACITOR, 0.1uF	MARK WITH 0.1uF OR SAME MEANING OF VALUE	YES
24	C5	CAPACITOR, 1000uF	MARK WITH 1000uF OR SAME MEANING OF VALUE	YES
25	C6	CAPACITOR, 0.1uF	MARK WITH 0.1uF OR SAME MEANING OF VALUE	YES
26	C7	CAPACITOR, 220uF	MARK WITH 220uF OR SAME MEANING OF VALUE	YES
27	INPUT	AS INPUT POWER	FIGURE 7	NOTE 2
28	OUTPUT A	AS OUTPUT POWER	FIGURE 7	NOTE 2
29	OUTPUT B	AS OUTPUT POWER	FIGURE 7	NOTE 2
30	U1	DIP 8 SOCKET	8 LEGS	NO
31	ON THE TOP OF ITEM 30	IC, LM358	8 LEGS	FIGURE 4
32	SWITCH	SLIDE SWITCH	SIX LEGS	FIGURE 10

NOTE 1: The maximum input voltage (Or you can say the maximum output voltage for each channel) for this kit is around 25V (I assume the PN junction at some components do not "eat" at voltage). This is known from the value of C1 and C7. If the input voltage is more than 25V, the capacitor would burn away first. Of course, if the capacitor of C1 and C7 we provide is 50V (We may use other higher value). The maximum input voltage would be 32V; this is the maximum input voltage of LM358. Other components are not needed to care because they still have large amount of space to withstand the energy consume at such component very easy even this is at 32V.
NOTE 2: You can say there are two directions to solder this component. Both are work but this is better the terminal is facing outside the PCB.

CIRCUIT DIAGRAM:

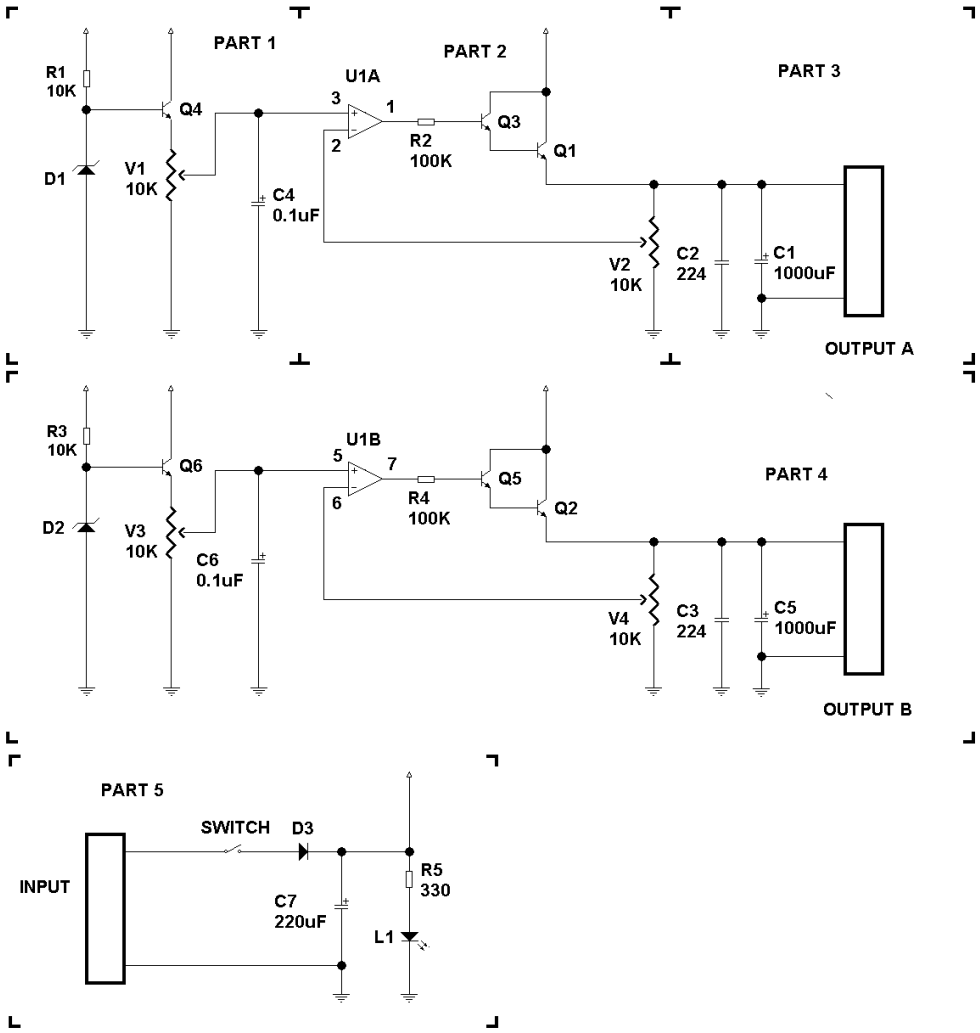

FIGURE 12