College of Engineering
Ateneo de Naga University

POWER SUPPLY

DOCUMENTATION AND PROCEDURE

Project by:

PAMELA KAYE C. MAGNIFICO
QCPA222-GO21

Submitted to:

ANACLETO CASIMIRO
Instructor

CASING

The specifications for the casing are referenced from the height of the transformer to be used, which is approximately 7 cm.

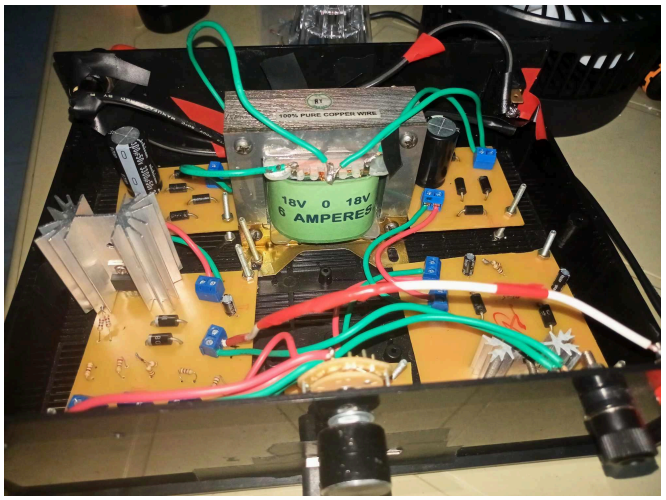
External: 25.3cm x 19cm x 8.15cm *Internal: 24.8cm x 17.75cm x 7.45cm*

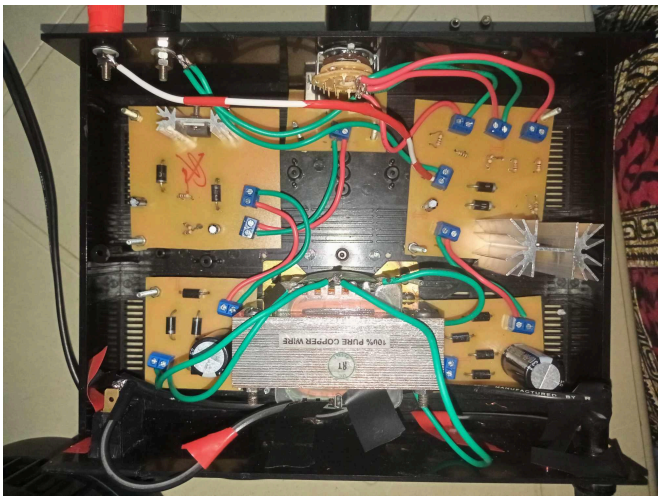
OUTSIDE CASING

FRONT

BACK

INSIDE CASING

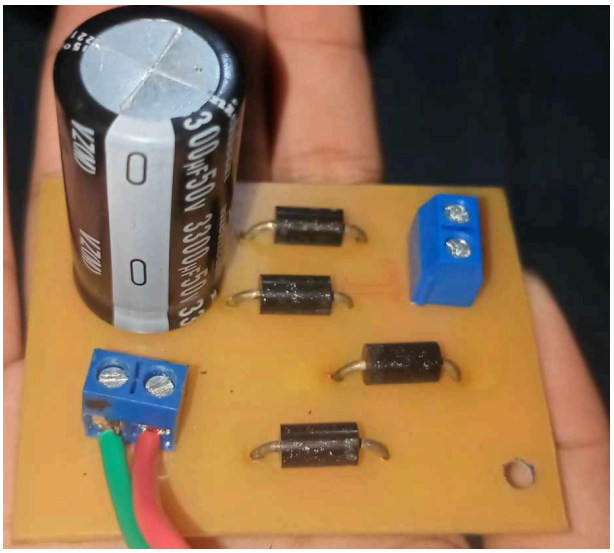


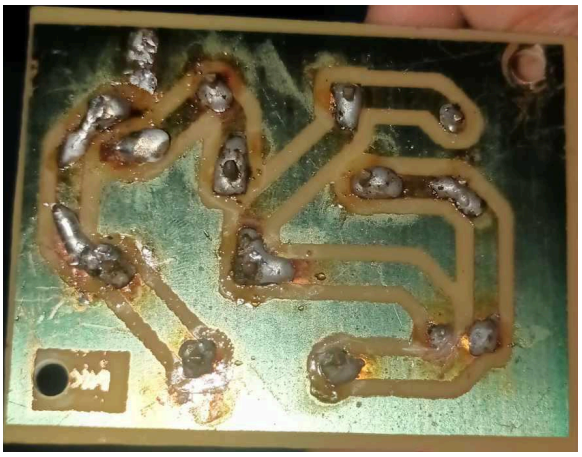


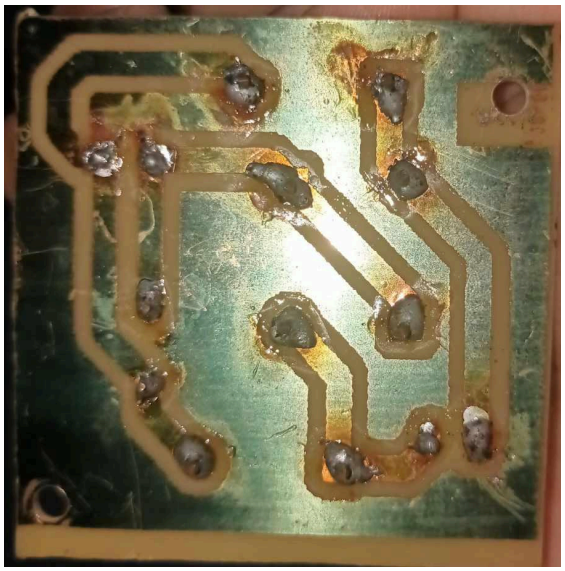
Note: The students opted to buy this casing due to unavailable stock of a better-sized plastic enclosure.

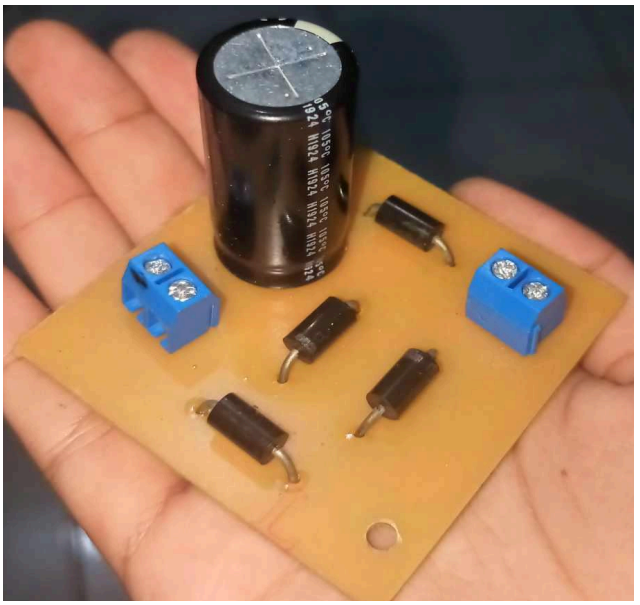
ACTUAL PCB

MODULE 1. FULL-WAVE BRIDGE RECTIFIER

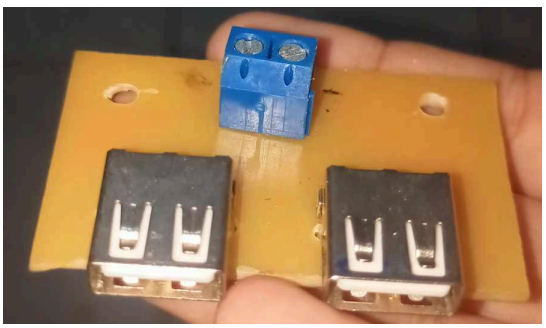




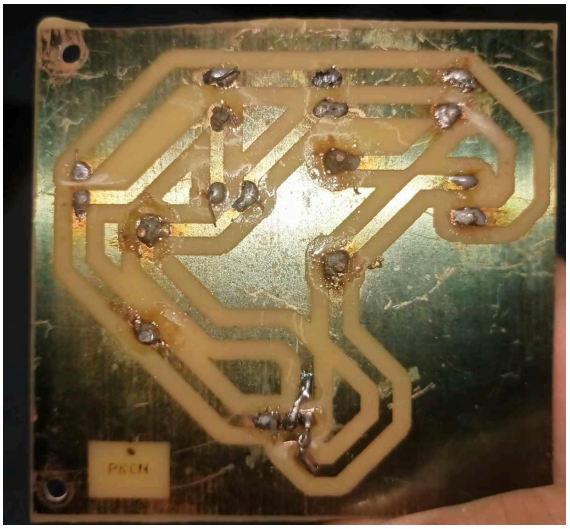


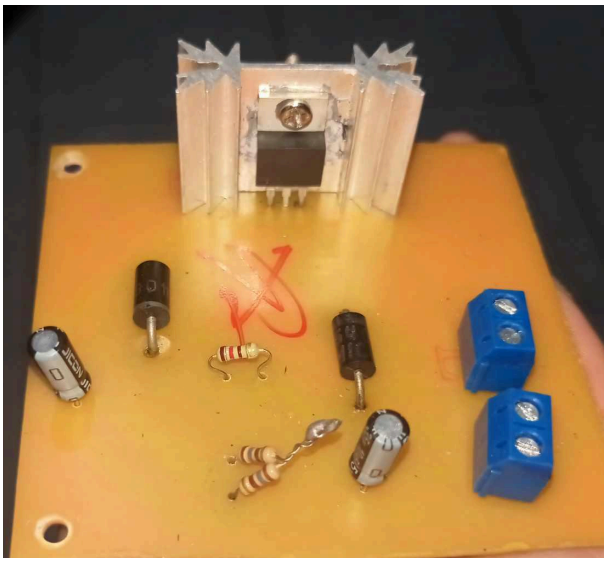


MODULE 4. USB PORT

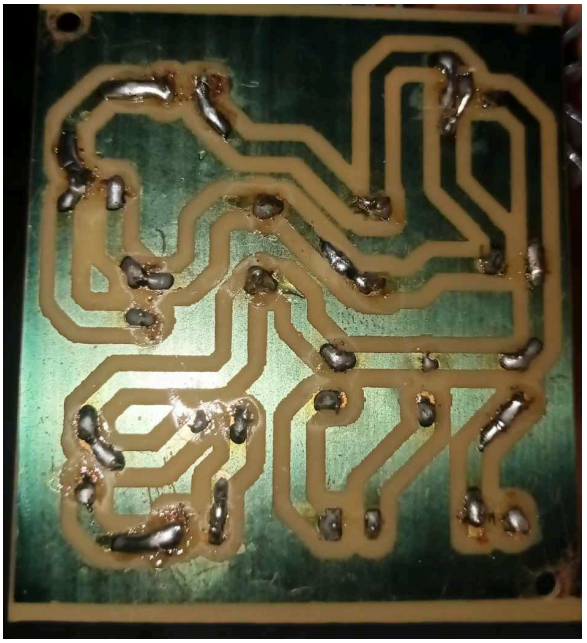


MODULE 2. FIXED 5V OUTPUT VOLTAGE REGULATOR





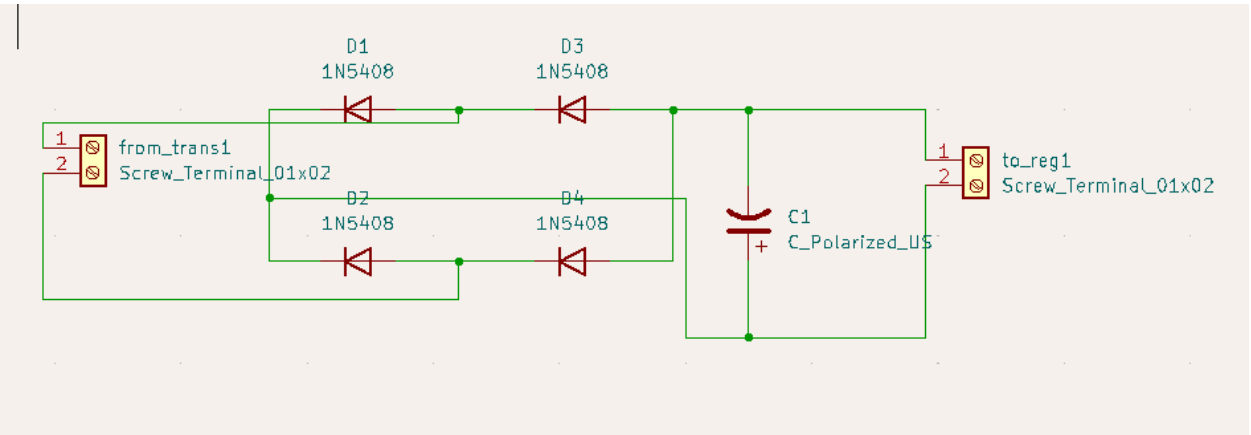
MODULE 3. VARIABLE REGULATOR



SCHEMATIC DIAGRAM

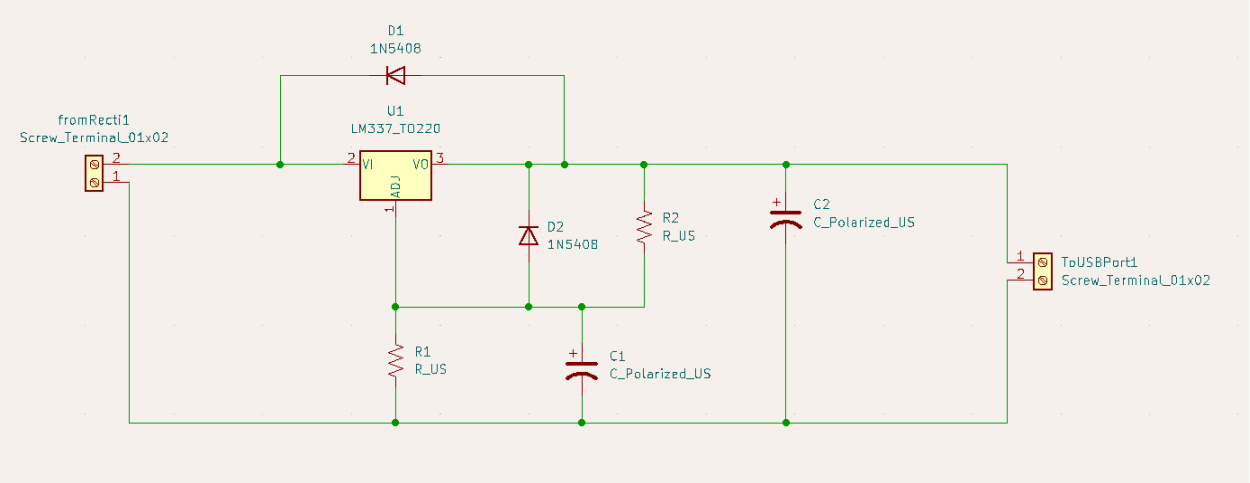
The schematic diagrams were created using KiCad 6.0 and were initially supplied by the instructor, with additional explanations and clarifications provided by the laboratory custodians. These diagrams were subsequently used to design PCBs, a feature of the aforementioned software. The following are the schematic diagrams we have utilized:

MODULE 1. FULL-WAVE BRIDGE RECTIFIER



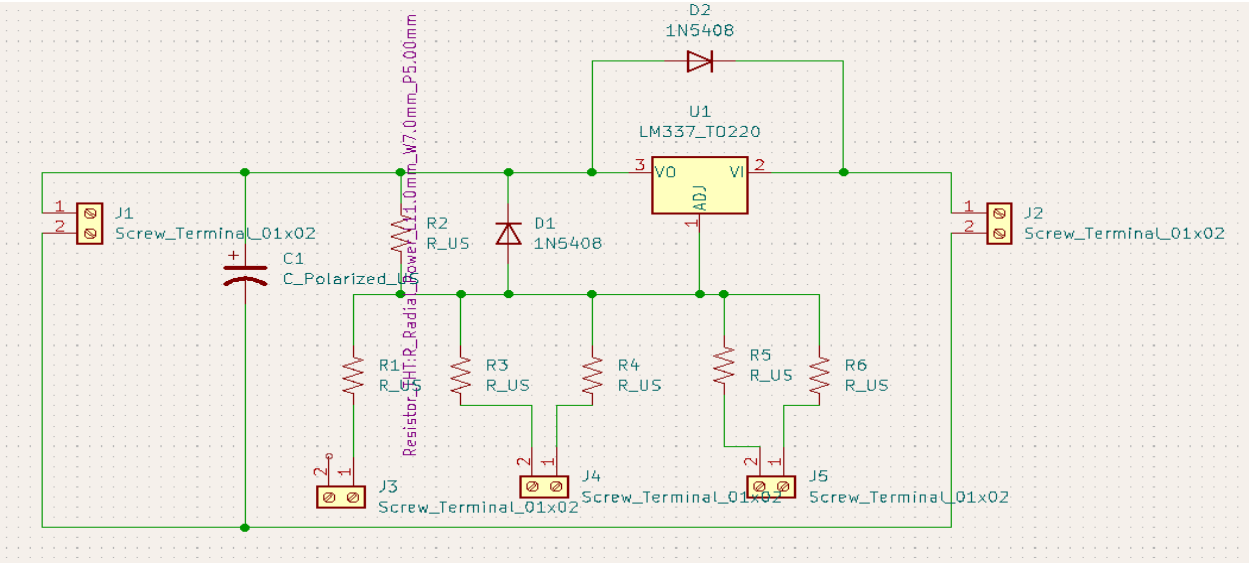
Note: This diagram was slightly modified due to an oversight, therefore is adjusted to still comprise similar connections with the correct one.

MODULE 2. FIXED 5V OUTPUT VOLTAGE REGULATOR



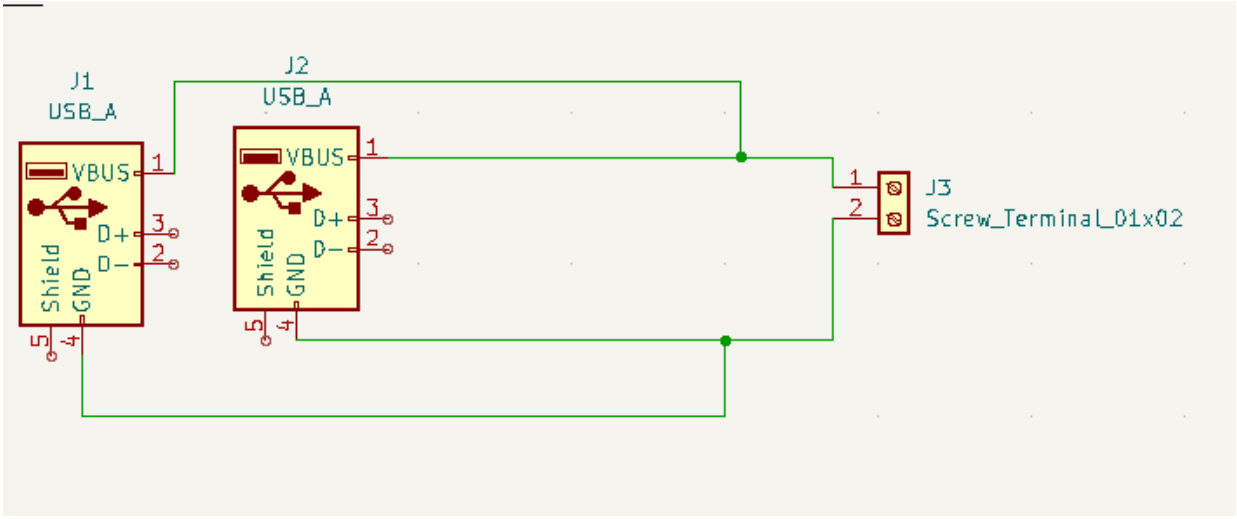
Note: The component used in this module is LM338, which has different pinning compared to LM337. However, the KiCad 6.0 software does not have any available footprints for LM338. Thus, LM337 is used as a substitute and the pins will be manually manipulated.

MODULE 3. VARIABLE REGULATOR



Note: The same issue with LM338 applies here. In addition, since the students have only utilized 2-pin terminal blocks and 5 expected voltage outputs, there is a pin without connection. It was supposedly reserved for the 0V placement in the rotary switch.

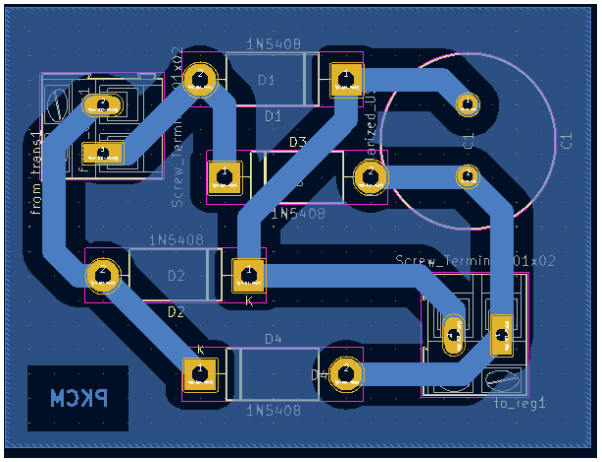
MODULE 4. USB PORT



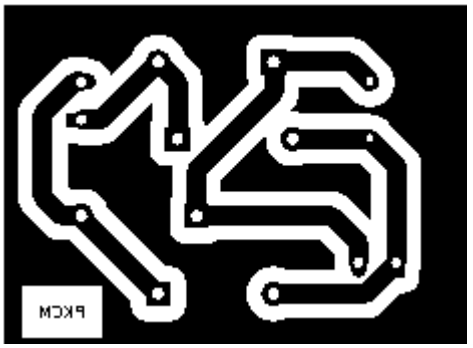
PCB DESIGN/LAYOUT

Using the schematic diagrams created in KiCad 6.0, PCB layouts are then designed and printed in a black and white format, which will be displayed below, as necessary, in the succeeding procedures. During the design process, the students made sure to estimate and reduce occupied space in consideration of the transformers and wiring. Additionally, the students were advised to avoid Y-shaped and complicated connections to avoid difficulty while soldering the components. Sufficient clearance was ensured to prevent the copper on the PCB from burning easily.

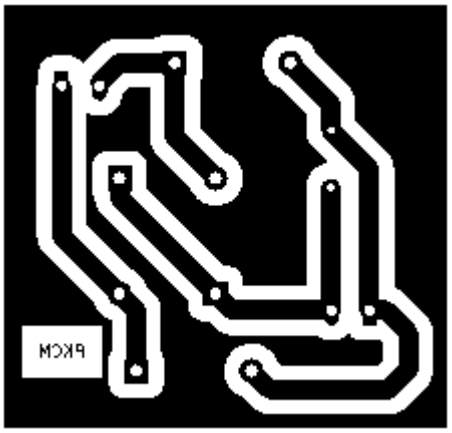
MODULE 1. FULL-WAVE BRIDGE RECTIFIER

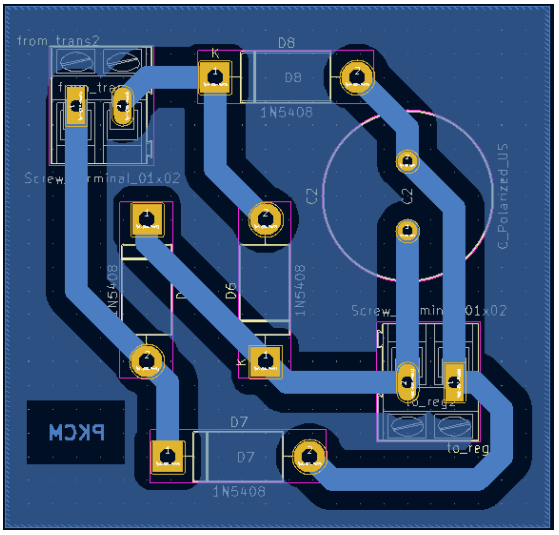


PRINT PREVIEW

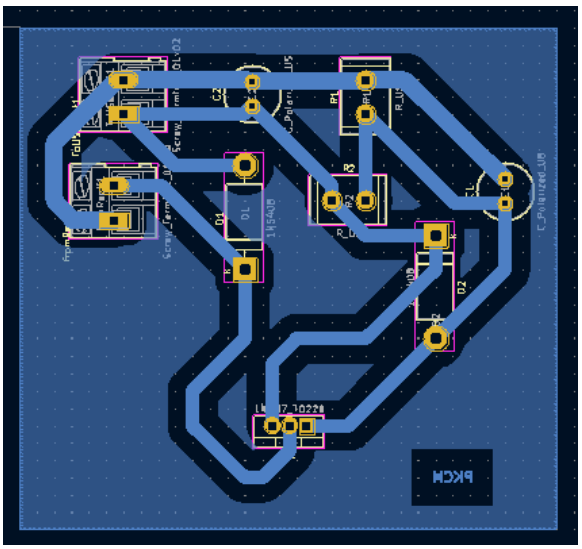


PRINT PREVIEW

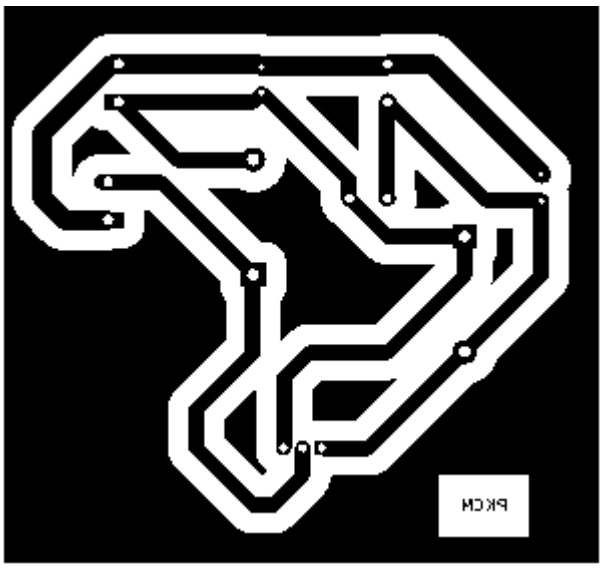




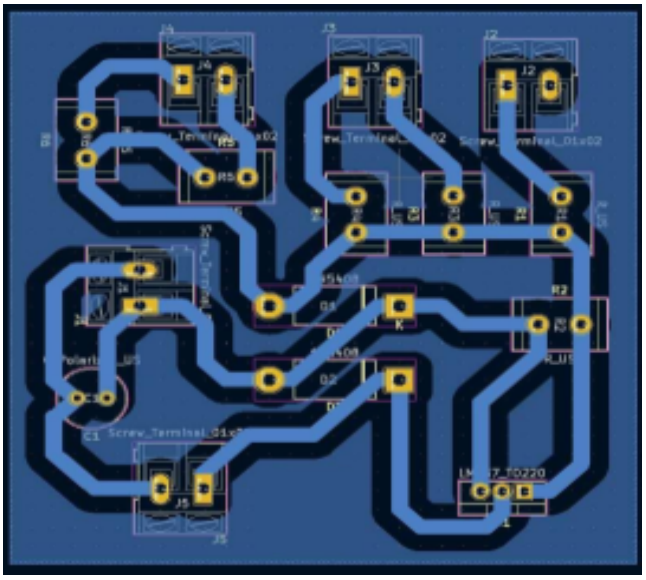
MODULE 2. FIXED 5V OUTPUT VOLTAGE REGULATOR



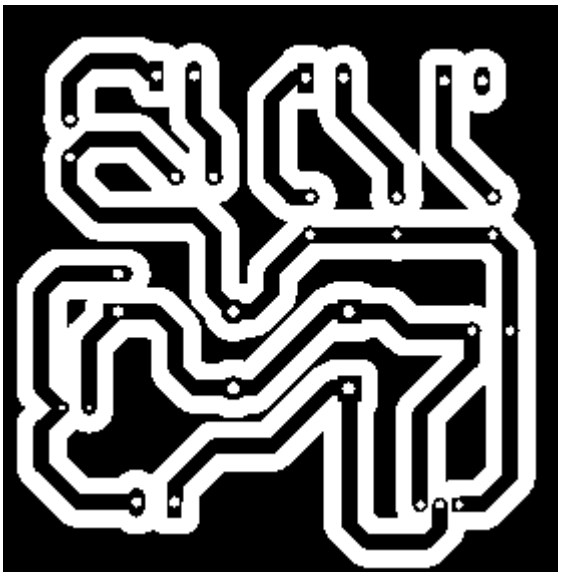
PRINT PREVIEW



MODULE 3. VARIABLE REGULATOR

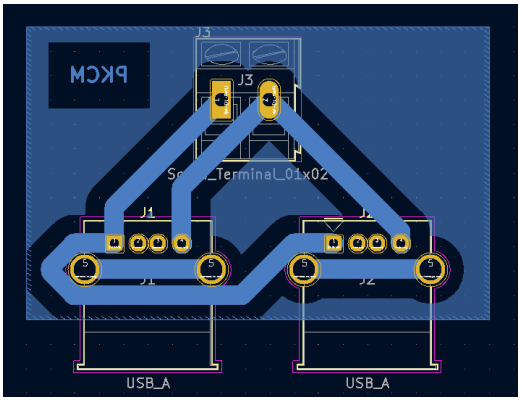


PRINT PREVIEW

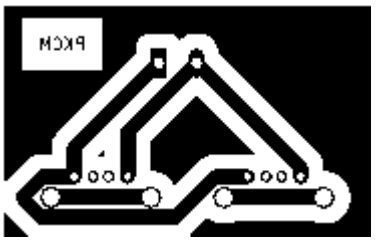


MODULE 4. USB PORT

Note:The USB footprint is intentionally placed outside since the component will be placed in a hole at the front of the casing.



PRINT PREVIEW

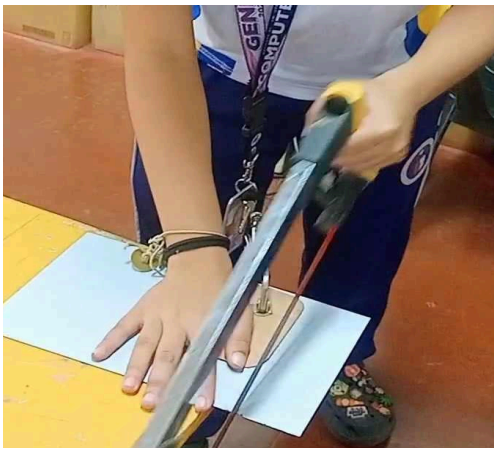


PROCEDURE

EXPOSURE AND ETCHING

- ***Cutting of PCB.*** The black and white print previews of the PCB layouts, previously shown, are printed precisely to a 1:1 scale. The PCB is then cut into specific sizes based on the printed layouts.

  *While acetate printing is available, it can also be costly and hard to acquire due to demand. Therefore, we had opted to print on ordinary bond paper and wet it using **cotton balls** with **baby oil** until the paper had the texture and opacity similar to a parchment or wax paper.*

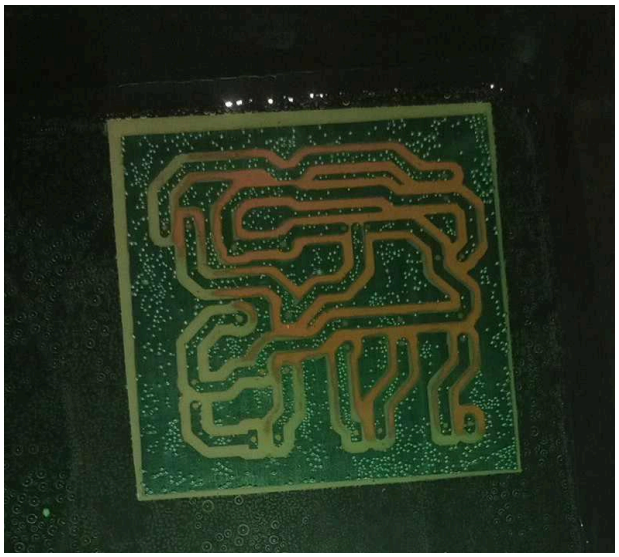


- After drying briefly, the papers are placed on top of the cut PCB for **6 minutes and 40 seconds** of exposure to laboratory UV light.

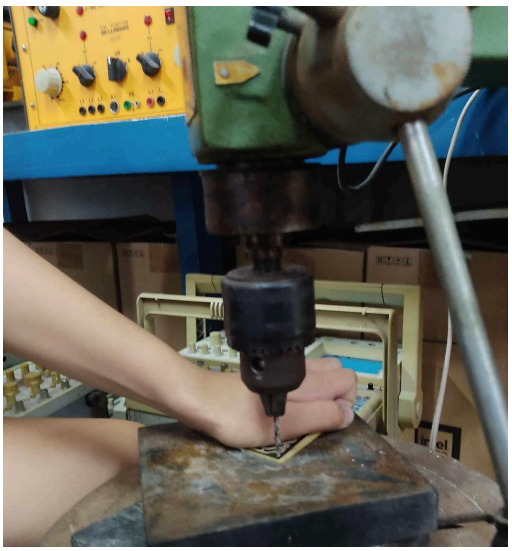
Meanwhile, the PCB developer powder is dissolved in 1 liter of water in a container. Following UV exposure, the PCB is immersed in the developer solution and agitated by shaking the container until the layout becomes visible.

- If successful, the PCB is washed and will proceed to etching. A mixture of hydrogen peroxide and 29% Muriatic Acid is placed in another container with a **2:1** ratio, respectively. Etching removes the unwanted copper in the circuit board and further clears out the exposed PCB layout.

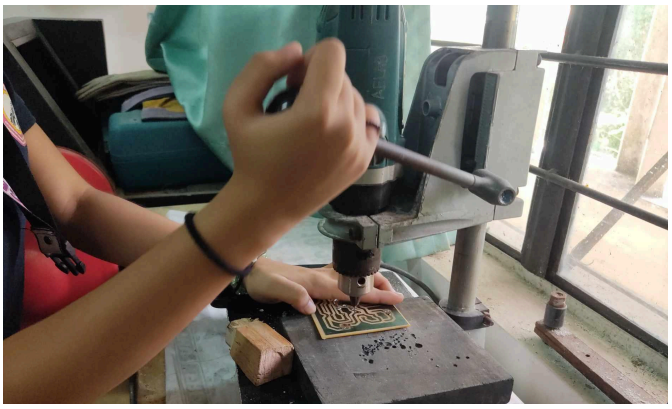
- The washed PCB is immersed and agitated until the copper in the board has dissolved.



DRILLING OF HOLES FOR COMPONENTS

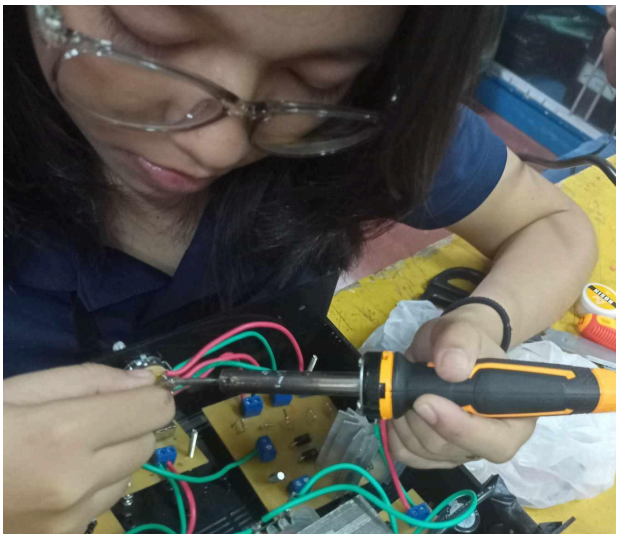


After etching, the holes for the components and screws were drilled. The available equipment in the workshop was utilized, as well as personally-owned drills and drill bits. Safety and precautions were practiced.



SOLDERING THE COMPONENTS

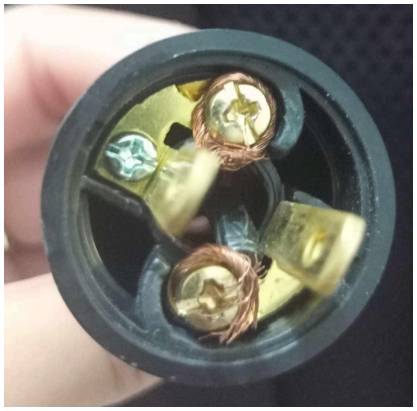
Referencing the schematic diagrams and PCB layouts, the available components were soldered into the printed circuit boards using a 40W Soldering Iron bought in bulk by the class. All the components can normally be soldered except for the LM338, which should habitually be rested while soldering in order to avoid overheating and short-circuiting. Continuity is verified through the use of a DMM.

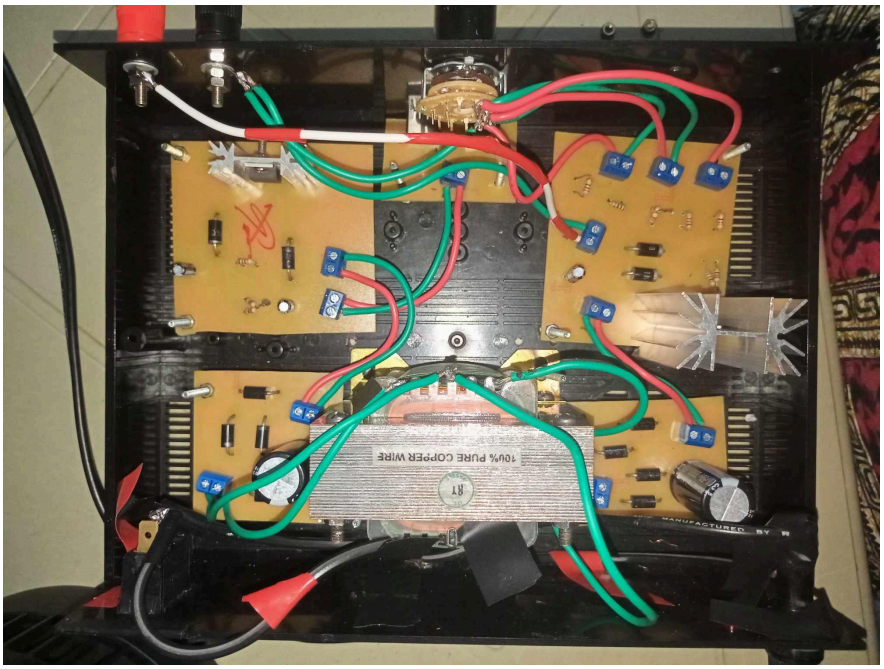


CASE FITTING AND DRILLING

After soldering the components, the PCBs and the transformer were then adjusted to fit inside the case to begin the drilling of holes for the screws, as well as to accommodate the fuse holder, plug head, ac plug, switch, usb port, and the rotary switch.

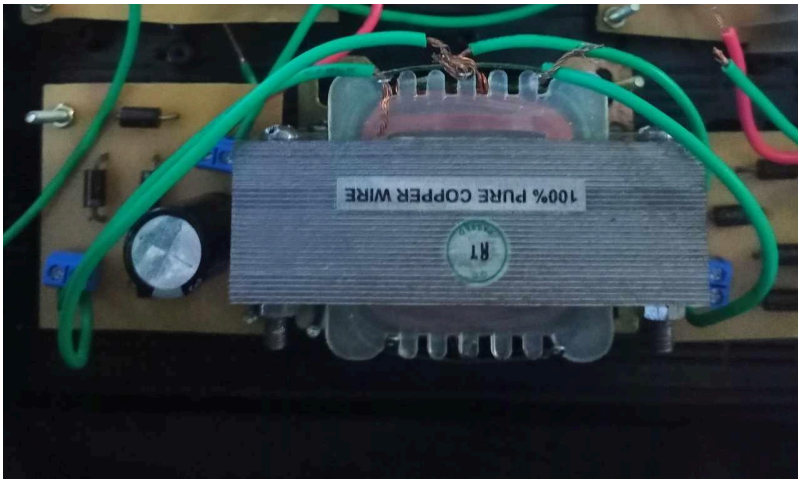
WIRING

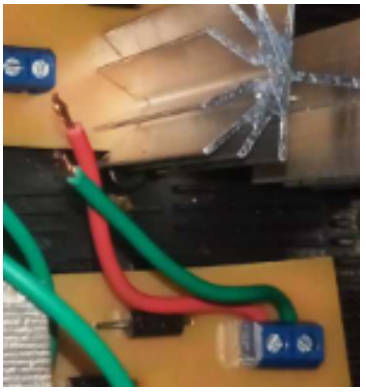




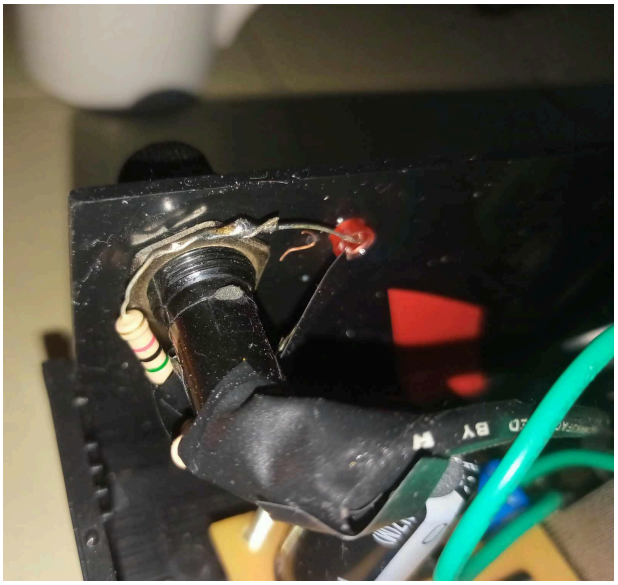
Two types of wires were used: Autowire #18 and an AC power plug. In the AC wire, its two stranded wires were separated: one to be connected to the switch to the 220V of the transformer, and the other to the fuse holder to the 0V of the transformer.

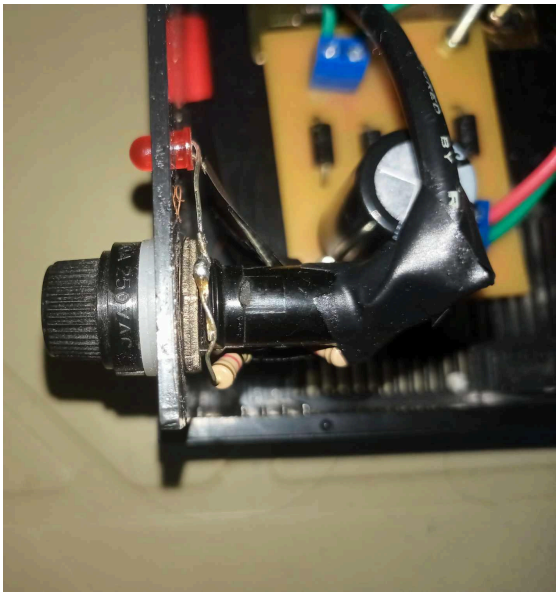
18V and 0V from the transformer were then connected to the input terminal of both rectifiers, using only GREEN autowire #18 to signify unconverted AC. In the rectifier output, however, red and green wires were used, red for positive and green for negative.





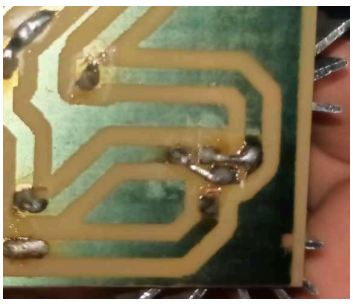
- The instructor required a Fuse Blown Indicator (FBI), which was done by placing two 5-kiloohm resistors, an LED,and the fuse holder in series, which is then connected to the plug. The fuse used is standard 1A.





- The respective output and input terminal blocks were connected: from rectifier to Module 2 to Module 4 (USB Port), and from rectifier to Module 3. In module 3, the positive pin of the output terminal block is connected to the red binding post, and the negative terminal to the black binding pos,t which is the ground. The respective pins of the terminal blocks for the variable output were soldered to the pins in the rotary switch, whose designated pole is also connected to the ground.

TROUBLESHOOTING



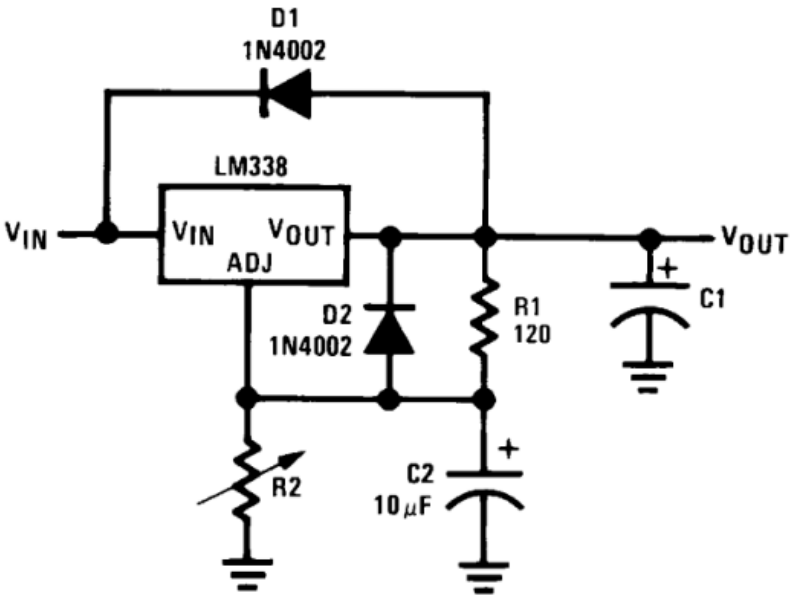
The input and output pins for the LM338 in Modules 2 & 3 were interchanged and re-soldered to accommodate the pinning of the LM337 footprint used in the PCB layout.

Furthermore, other components, such as the terminal blocks, were customarily re-soldered, as well as the connections in the PCB when disconnected. The connections are verified by using a DMM and checking for continuity.

Additionally, it is to note that the students only have foundational knowledge in electronics and thus are prone to novice mistakes during this project. They were assisted by their instructors and laboratory custodians.

APPENDICES

I. COMPUTATION FOR RESISTOR VALUES



$$V_{OUT} = 1.25V\left(1 + \frac{R2}{R1}\right) + I_{ADJ}R2$$

A. *MODULE 2. FIXED 5V OUTPUT VOLTAGE REGULATOR*

I_{ADJ} = 60µA R_1 = 120Ω

$$V_{out} = 1.25V\,(1 + \frac{R_2}{R_1}) + I_{ADJ}R_2)$$

$$(5V = 1.25V\,(1 + \frac{R_2}{120\Omega}) + (60\mu A)R_2)120\Omega$$

$$600V = 150V\Omega + 1.25VR_2 + (60\mu A)(120\Omega)R_2$$

$$600V - 150V\Omega = 1.25VR_2 + 7.2x10^{-3}VR_2$$

$$450V\Omega = R_2(1.25V + 7.2x10^{-3}V)$$

$$\frac{450V\Omega}{1.2572V} = R_2$$

$$R_2 = 357.93827\Omega$$

B. *MODULE 3. VARIABLE REGULATOR*

I_{ADJ} = 60μA R_1 = 360Ω

$$V_{out} = 1.25V\,(1 + \frac{R_2}{R_1}) + I_{ADJ}R_2)$$

$$V_{out} = 1.25V + 1.25V(\frac{R_2}{360\Omega}) + (60\mu A)R_2$$

$$V_{out} - 1.25V = R_2(\frac{1.25V}{360\Omega}) + (60\mu A)R_2$$

$$V_{out} - 1.25V = R_2(3.532222x10^{-3})$$

$$R_2 = (1.25V + 7.2x10^{-3}V)$$

$$\frac{V_{out} - 1.25V}{3.532222x10^{3}} = R_2$$

1. $V_{out} = 1.5V$

$$\frac{1.5V - 1.25V}{3.532222x10^{3}} = R_2 \qquad 70.77697\,\Omega = R_2$$

2. $V_{out} = 3V$

$$\frac{3V - 1.25V}{3.532222x10^{3}} = R_2 \qquad 495.43881\,\Omega = R_2$$

3. $V_{out} = 6V$

$$\frac{6V - 1.25V}{3.532222x10^{3}} = R_2 \qquad 1344.7625\,\Omega = R_2$$

4. $V_{out} = 9V$

$$\frac{9V - 1.25V}{3.532222x10^{3}} = R_2 \qquad 2194.08619\,\Omega = R_2$$

5. $V_{out} = 12V$

$$\frac{3V - 1.25V}{3.532222x10^{3}} = R_2 \qquad 3043.4098\,\Omega = R_2$$

II. COSTING

COMPONENTS	PRICE PER UNIT	QUANTITY	TOTAL PRICE	SOURCE
CAPACITOR 3300uF 50V x 5	550	1	550	RSPH
LM338T x 2	340.07	1	340.07	RSPH
PCB	335	1	335	LAZADA
TAMAGAWA TRANSFORMER 6A 18V-0-18V	965	1	965	LAZADA
SOLDERING IRON	432.5	1	432.5	SHOPEE
AUTOWIRE #18	28	4	112	3GX
TERMINAL BLOCK 2 PIN & ROTARY SWITCH	91.15	1	91.15	SHOPEE
CASING	330	1	330	LAZADA
USB PORT	32.15	1	32.15	SHOPEE
DIODES	10	16	160	3GX
CAPACITOR 10uF 25V	5	3	15	3GX
BINDING POST	21	2	42	3GX
AC PLUG	40	1	40	3GX
PLUG HEAD	30	1	30	3GX
FUSE 1A	3	3	9	3GX
FUSE HOLDER	18	1	18	3GX
SWITCH (3 pin)	29	1	29	3GX
RESISTORS	1	13	13	3GX
LEAD SOLDER AND ROSIN	93	1	93	YASHANO
MURIATIC ACID	16.14283	1	16.14283	YASHANO
DESOLDERING PUMP	6.29	1	6.29	YASHANO
HYDROGEN PEROXIDE	12.50	1	12.50	YASHANO
BOLTS AND NUTS	3.00	20	60.00	La Suerte
NUTS (EXTRA)	1.00	20	20.00	La Suerte
GRAND TOTAL (shipping included)	**3751.79853**			