REDHAWK LABORATORY POWER SUPPLY—CIRCUIT BOARD DESIGN

This guide focuses on quickly designing a basic 120VAC to DC power supply. It is not intended to teach KiCAD. Students should use this guide alongside the official KiCAD documentation. For detailed KiCAD instructions, visit [Getting Started in KiCad | 8.0 | English | Documentation | KiCad]

1. DOWNLOADING AND INSTALLING KICAD

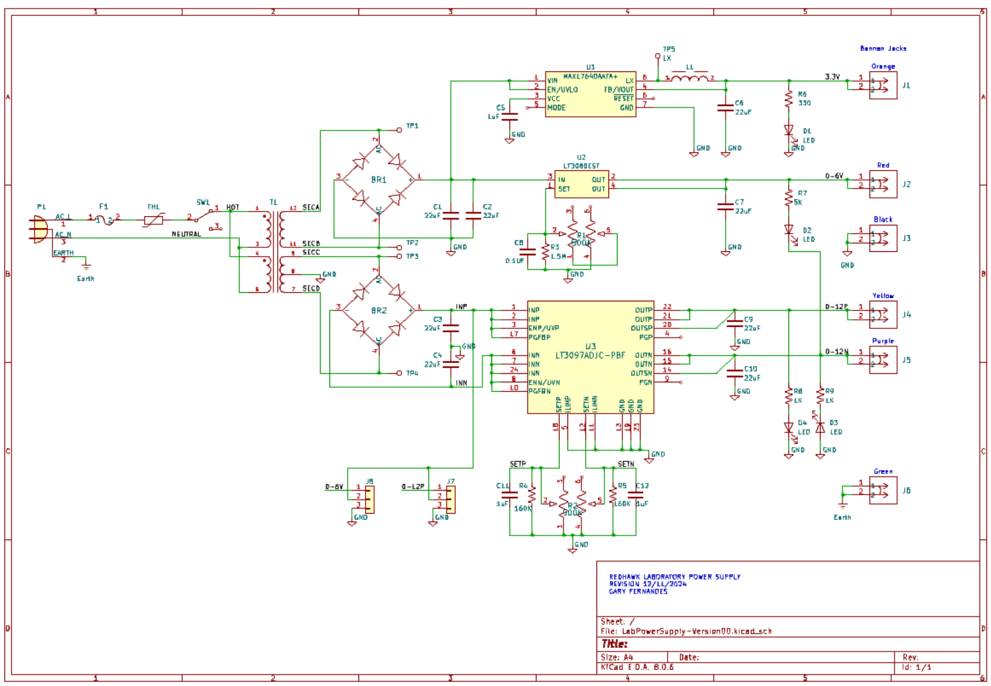
For those running on personal devices. KiCad runs on many operating systems, including Microsoft Windows, Apple macOS, and many major Linux distributions. You can find the most up to date instructions and download links at https://www.kicad.org/download/ .

2. PCB DESIGN—BASIC CONCEPTS AND WORKFLOW

The typical workflow in KiCad consists of two main tasks: drawing a schematic and laying out a circuit board.

The schematic is a symbolic representation of the circuit: which components are used and what connections are made between them.

Schematic



The board is the physical realization of the schematic, with component footprints positioned on the board and copper tracks making the connections described in the schematic. Footprints are a set of copper pads that match the pins on a physical component. When the board is manufactured and assembled, the component will be soldered onto its corresponding footprint on the circuit board.

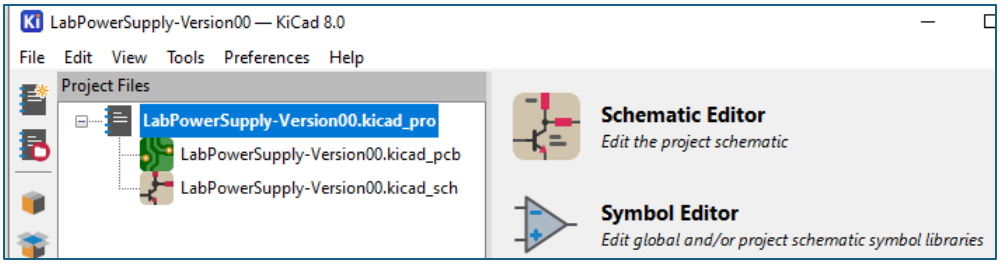
PCB Layout

<Lab power supply example layout>

KiCad uses separate app windows for drawing the schematic ("Schematic Editor"), laying out the board ("PCB Editor"), and editing symbols and footprints ("Symbol Editor" and "Footprint Editor").

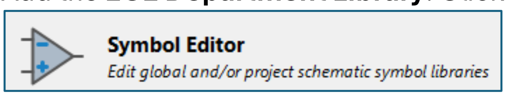
3. PROJECT CREATION

The first thing to do when starting a new design is to create a new project. Opening KiCad will bring up the Project Window. Click **File → New Project**, browse to your desired location (**suggest using your OneDrive**), and give your project a name such as **LaboratoryPowerSupply-Version00**. Make sure the ***Create a new folder for the project* checkbox** is check marked, then click Save. This will create your project files in a new subfolder with the same name as your project.

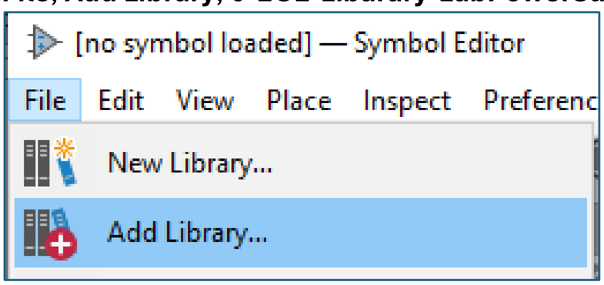


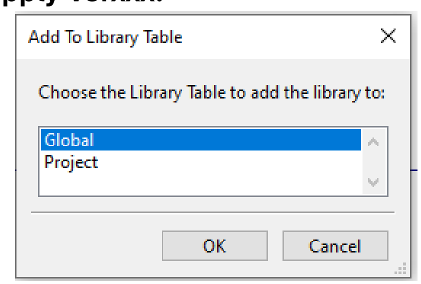
4. CUSTOM SYMBOLS AND FOOTPRINTS LIBRARY

4.1. Add the **ECE Department Library**. Click on the ***Symbol Editor*** icon.

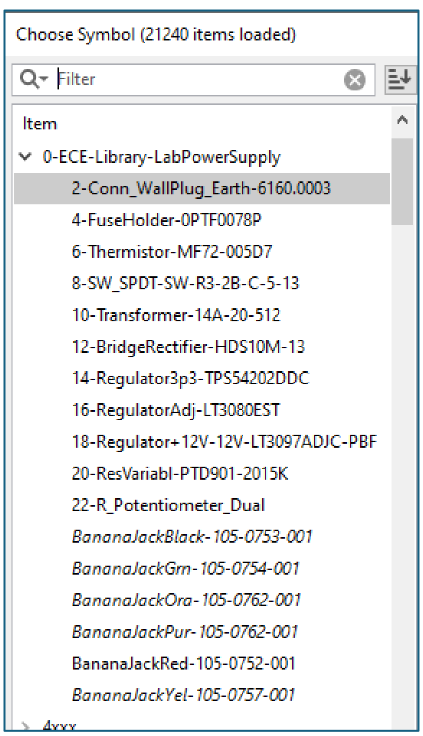


4.2. **File, Add Library, 0-ECE-Libarary-LabPowerSupply-verxxx.**





4.3. The ECE part library should now be available:



WHAT AM I LEARNING HERE? THAT'S UP TO YOU!

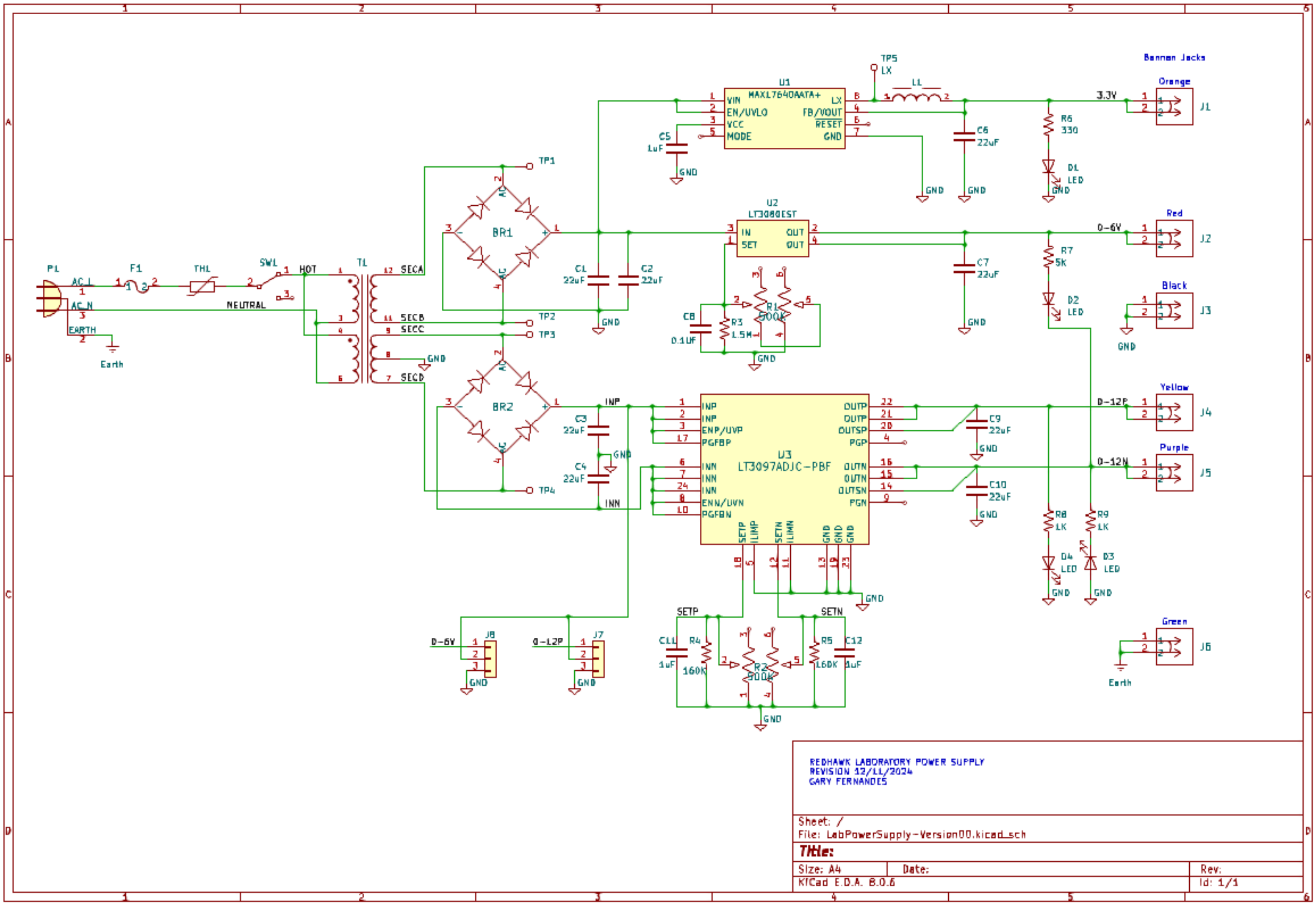
Think of datasheets as engineering scripture—master datasheets to avoid the blazing inferno of design failure. A well-understood datasheet ensures you won't end up feeling crucified by your own circuitry.

When designing this power supply, you have two approaches: you can rush through this document step-by-step to a quick completion, retaining little for your on-the-job future—or you can take the time to be curious and try to understand each component more. As a future circuit designer, it's essential to either have or develop curiosity!, fascination!, and self-drive to fully understand the components you're using, and this is where mastering datasheets makes the difference. Embrace datasheets as fascinating resources, not just technical documents. You know you're evolving into a real circuit designer when you find yourself binge watching YouTube tutorials, exploring how the electronic parts you are working with in the labs operate, whether for just 3 minutes or a full hour. Try it! The YouTube deep dives provide knowledge far beyond textbooks or this quick board design session.

And know that you don't need to be a circuit designer to find a fulfilling role in the electrical and computer engineering field; our discipline is vast and offers countless paths you may be drawn to.

5. ADDING SYMBOLS TO THE SCHEMATIC

Start building the schematic as shown below by clicking the **Add Symbol** button on the right side of the window. Initially, your schematic will only have scattered unwired parts.



An extra-large copy of the schematic is located at the end of this document.

> *DUAL MONITORS: As an engineer, using dual monitors will enhance your speed and accuracy. Keep KiCAD open on one screen and follow the steps below on the other screen.*

PART 1. 120VAC POWER RECEPTACLE

Place the following part, refer to the schematic for placements.

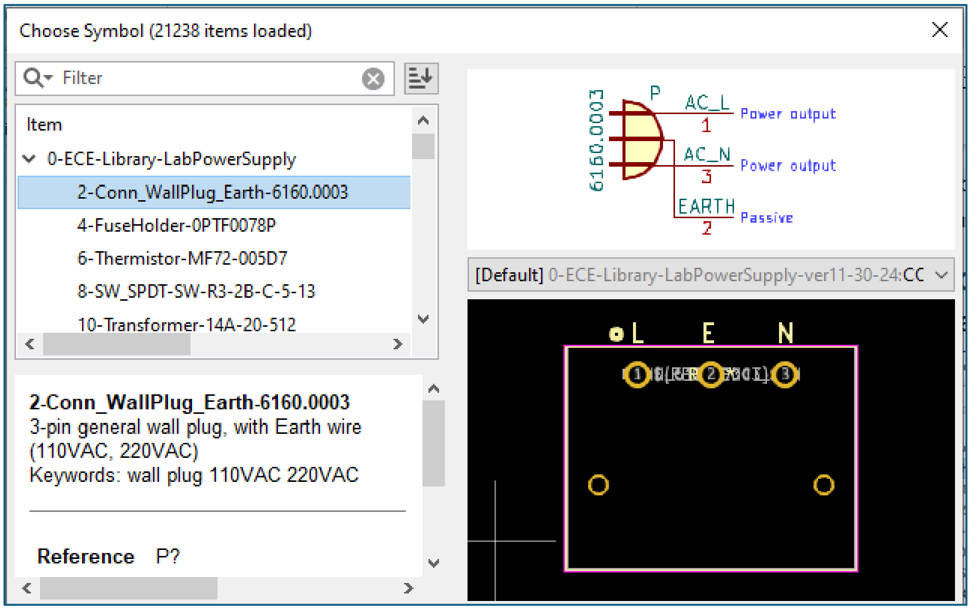
Quantity: 1

Wall Plug: 6160.0003 https://www.digikey.com/short/8zb5hz48

> *SUGGESTION: Click the datasheet link to briefly review the physical component. You might recognize it from products you've used before. Take a moment to read about it to better understand what you're about to build! Get familiar with using parts distributors, such as Digikey.*

Schematic Symbol: See pic below.

Footprint: See pic below.



PART 2. FUSE HOLDER

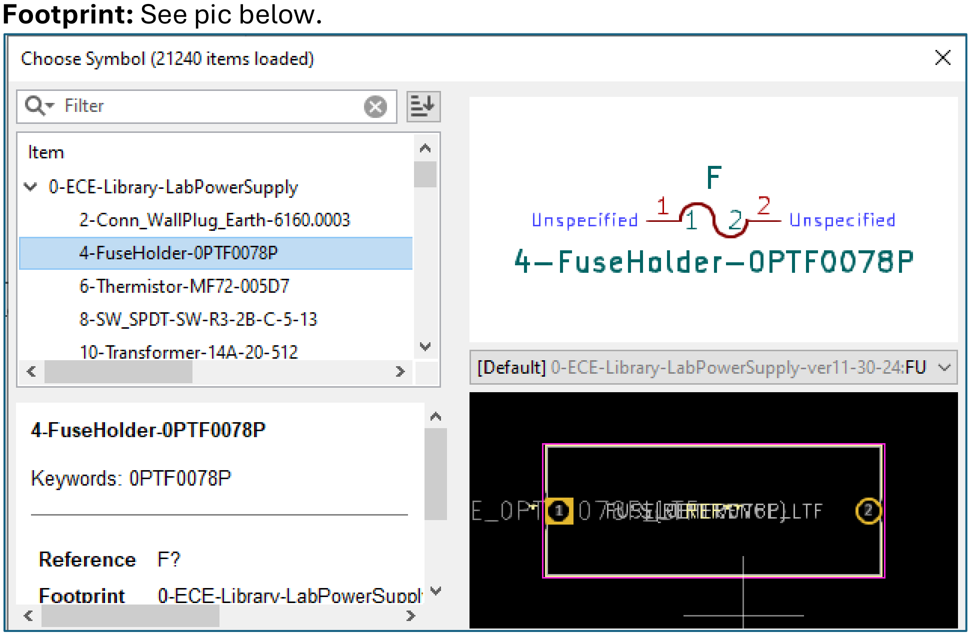
Place the following part, refer to the schematic for placements.

Quantity: 1

Fuse Holder: 0PTF0078P https://www.digikey.com/short/r7q1mz1n

Schematic Symbol: See pic below.

Footprint: See pic below.



PART 3. THERMISTOR (OPTIONAL)

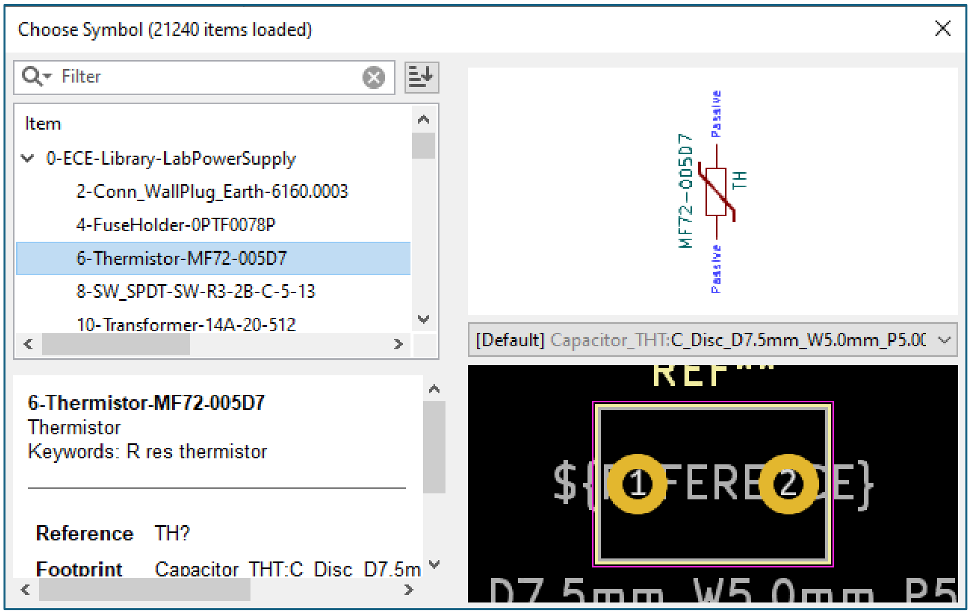
The thermistor has been included solely for your reference and understanding of its use in power supply design.

Quantity: 1

Thermistor: MF72-005D7 https://www.digikey.com/short/20545hj7

Schematic Symbol: See pic below.

Footprint: See pic below.



Thermistors are used in power supplies for the following purposes:

A. Limiting inrush current: A negative temperature coefficient (NTC) thermistor is in series with the supply line to limit inrush current in AC-DC power supplies.
B. Temperature-tracking: Thermistors can be used to track temperature in power supplies.
C. Inrush current limiter: They present higher resistance initially to prevent large currents during turn-on, and then heat up to allow higher current flow during normal operation.

PART 4. POWER SWITCH

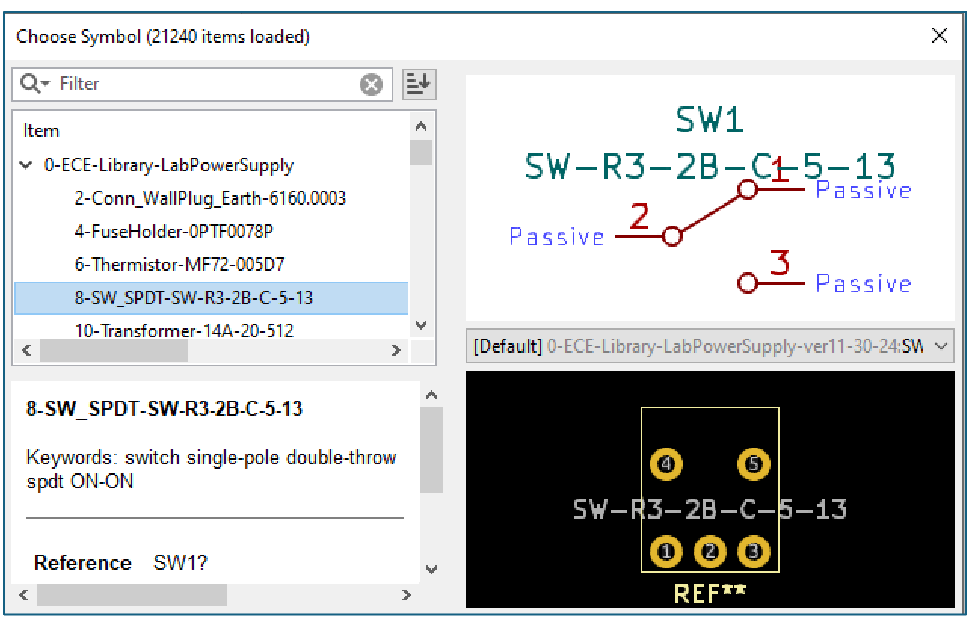
Place the following part, refer to the schematic for placements.

Quantity: 1

Switch: SW-R3-2B-C-5-13 https://www.digikey.com/short/q4b8h972

Schematic Symbol: See pic below.

Footprint: See pic below.



PART 5. TRANSFORMER

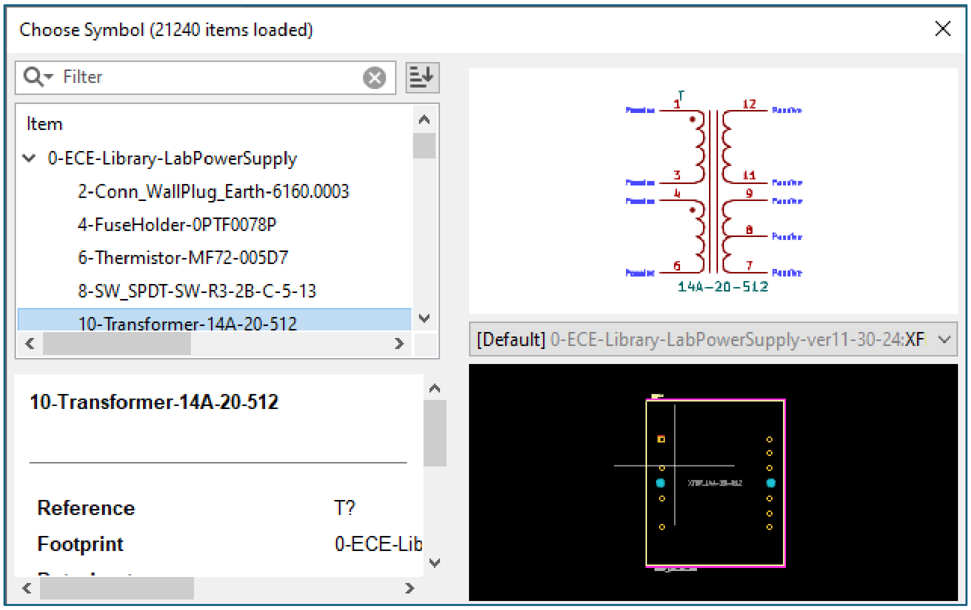
Place the following part, refer to the schematic for placements.

Quantity: 1

Transformer: 14A-20-512 https://www.digikey.com/short/h02rjm33

Schematic Symbol: See pic below.

Footprint: See pic below.



OPTIONAL: From the datasheet, here's a question for you: What does this mean? Feel free to ask us!

Part Number	VA Size
14A-20-512	20

PART 6. BRIDGE RECTIFIER

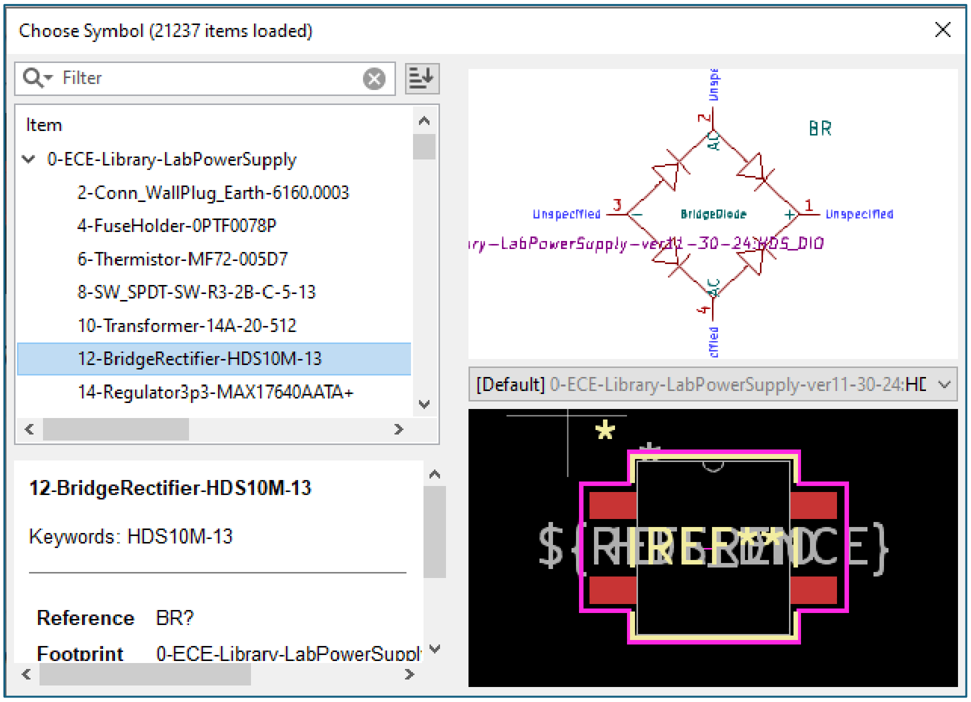
Place the following parts, refer to the schematic for placements.

Quantity: 2

Bridge Rectifier: HDS10M-13 https://www.digikey.com/short/v3h35qnq

Schematic Symbol: See pic below.

Footprint: See pic below.



PART 7. 3.3V DC/ DC CONVERTER REGULATOR

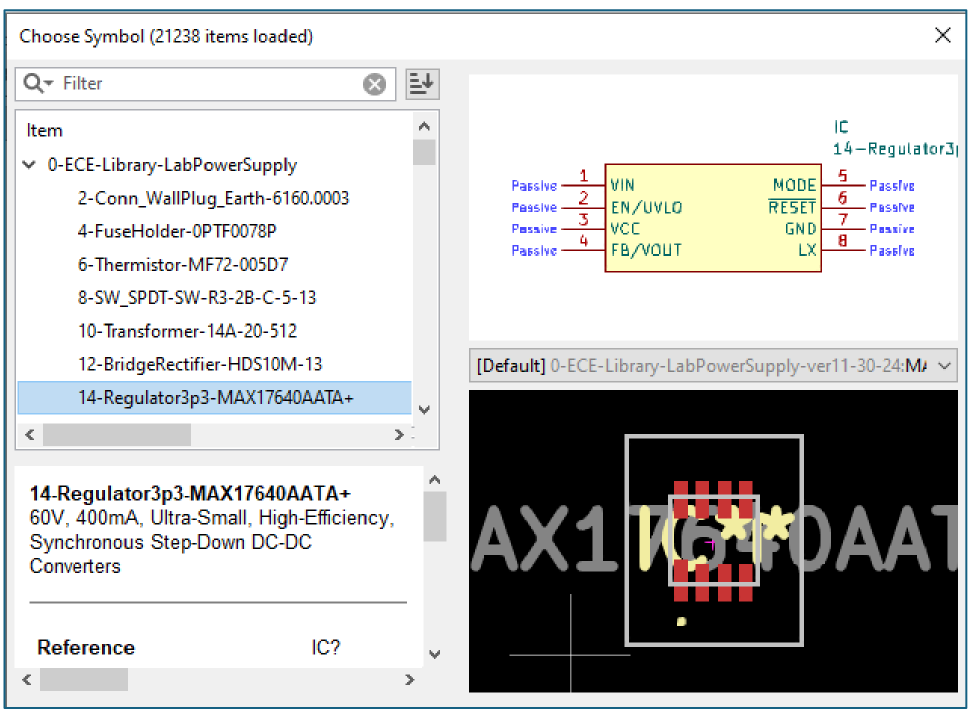
Place the following part, refer to the schematic for placements.

Quantity: 1

DC/DC Converter: MAX17640AATA+T https://www.digikey.com/short/r9q59wmb

Schematic Symbol: See pic below.

Footprint: See pic below.



OPTIONAL: From the datasheet, here's a question for you: What is the max wattage used? Feel free to ask us!

PART 8. 0-5.45VDC ADJUSTABLE LINEAR REGULATOR

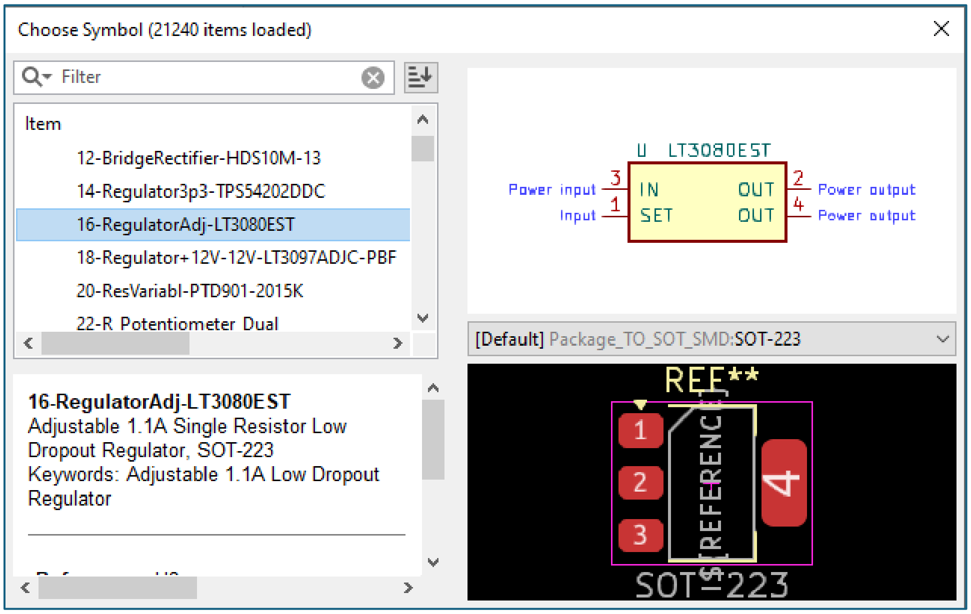
Place the following part, refer to the schematic for placements.

Quantity: 1

Linear Regulator: LT3080EST#PBF https://www.digikey.com/short/jnb7phtf

Schematic Symbol: RegulatorAdj-LT3080EST

Footprint: Package_TO_SOT_SMD:SOT-223 (be sure to select this from the dropdown menu)



OPTIONAL: From the datasheet above and below, can you determine the combined maximum wattage used by the three regulators? When you combine all three sub-power circuits fed by one transformer, you certainly don't want to risk it catching on fire! Feel free to ask us!

PART 9. ±0-12VDC DUAL LINEAR REGULATOR

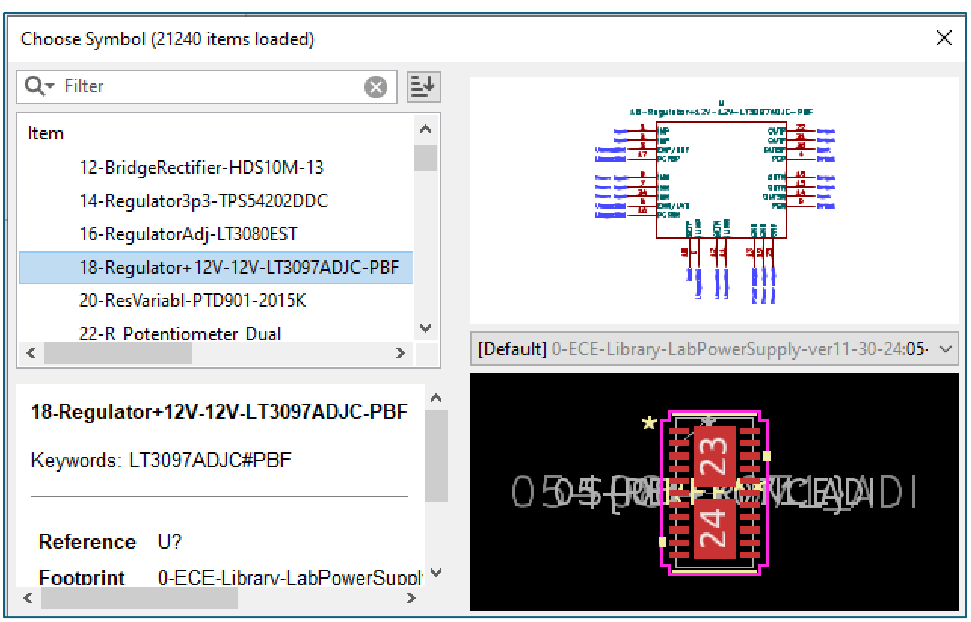
Place the following part, refer to the schematic for placements.

Quantity: 1

Dual Linear Regulator: LT3097ADJC#PBF https://www.digikey.com/short/pqvzrb2t

Schematic Symbol: See pic below.

Footprint: See pic below.



PART 10. POTENTIOMETER

Place the following parts, refer to the schematic for placements. NEUTRAL

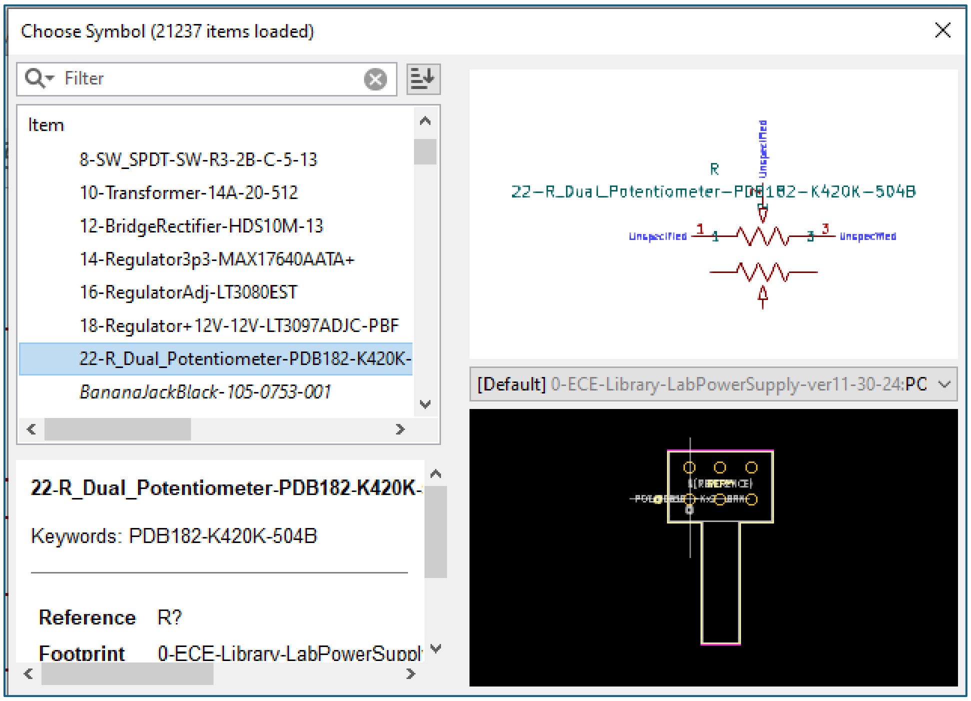
200K Ohm Dual Potentiometer Resistors: PDB182-K420K-204B https://www.digikey.com/short/8wb307vb

500K Ohm Dual Potentiometer Resistors: PDB182-K420K-504B https://www.digikey.com/short/8wb307vb

Quantity: 2

Schematic Symbol: 22-R_Dual_Potentiometer-PDB182-K420K-504B

Footprint: 0-ECE-Library-LabPowerSupply-ver11-30-24:POT_PDB182-Kx20_BRN



PART 11. INDUCTOR

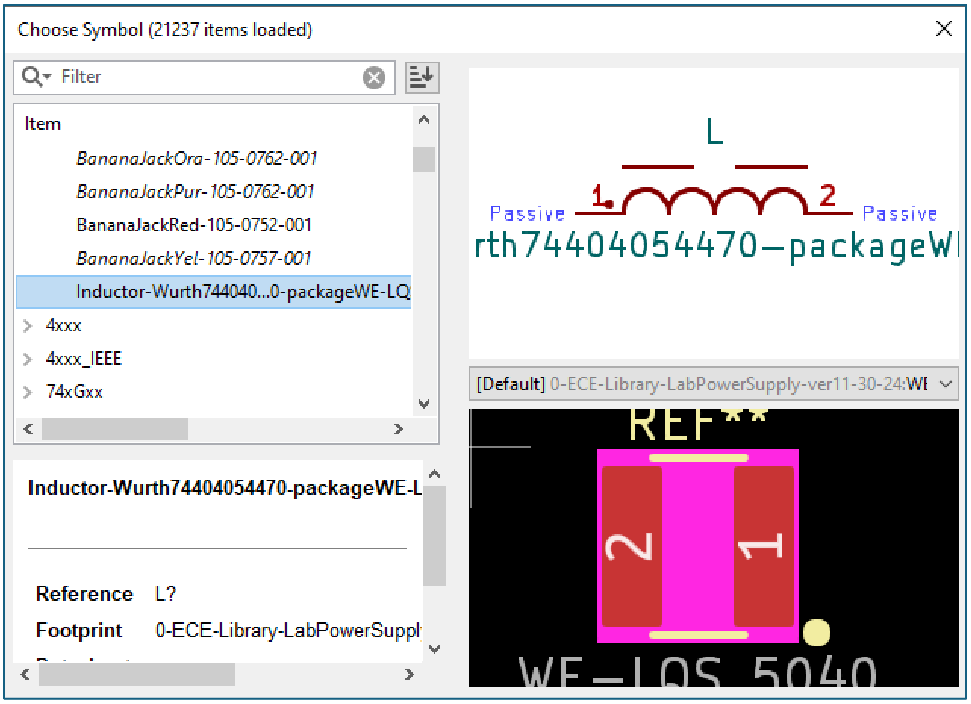
Place the following part, refer to the schematic for placements.

47uH Inductor: 74404054470 https://www.digikey.com/short/1rw1mr58

Quantity: 1

Schematic Symbol: Inductor-Wurth74404054470-packageWE-LQS_5040

Footprint: 0-ECE-Library-LabPowerSupply-ver11-30-24:WE-LQS_5040



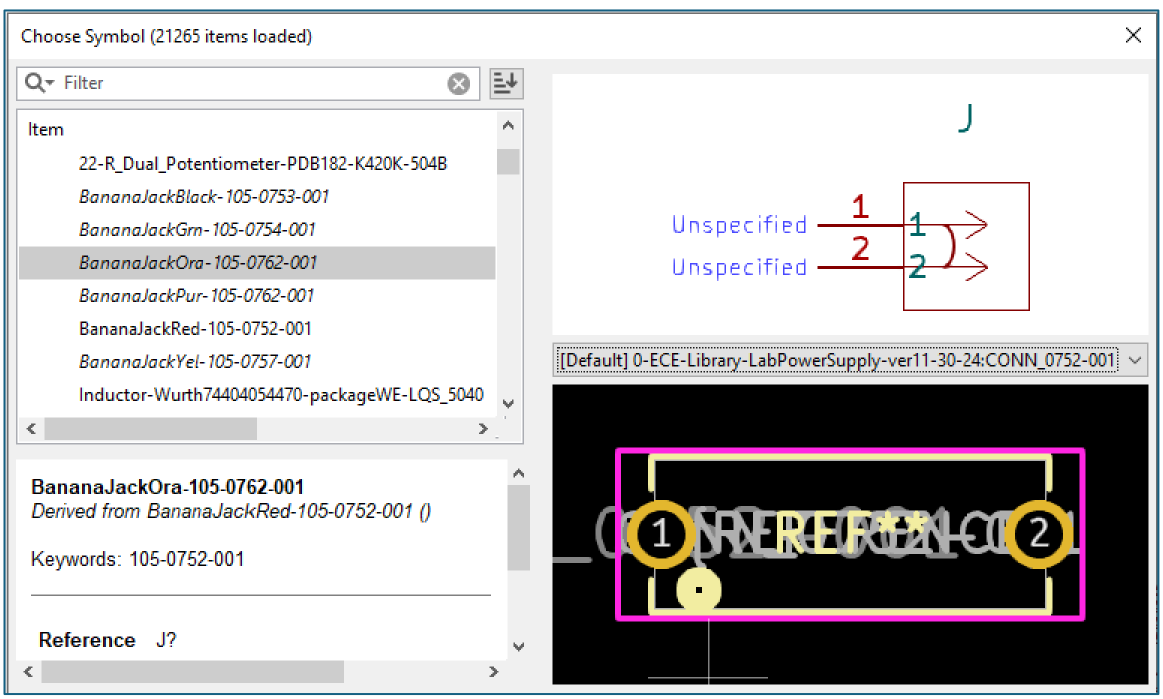
PART 12. OUTPUT POWER JACKS

Place a banana jack for the following power outputs:

Orange: 3.3Vdc. **Red**: 0-6Vdc. **Black**: PowerGND. **Yellow**: +12Vdc. **Purple**: -12Vdc. **Green**: Earth GND.

Schematic Symbol: See pic below. Okay to use same part for all six jacks.

Footprint: See pic below.

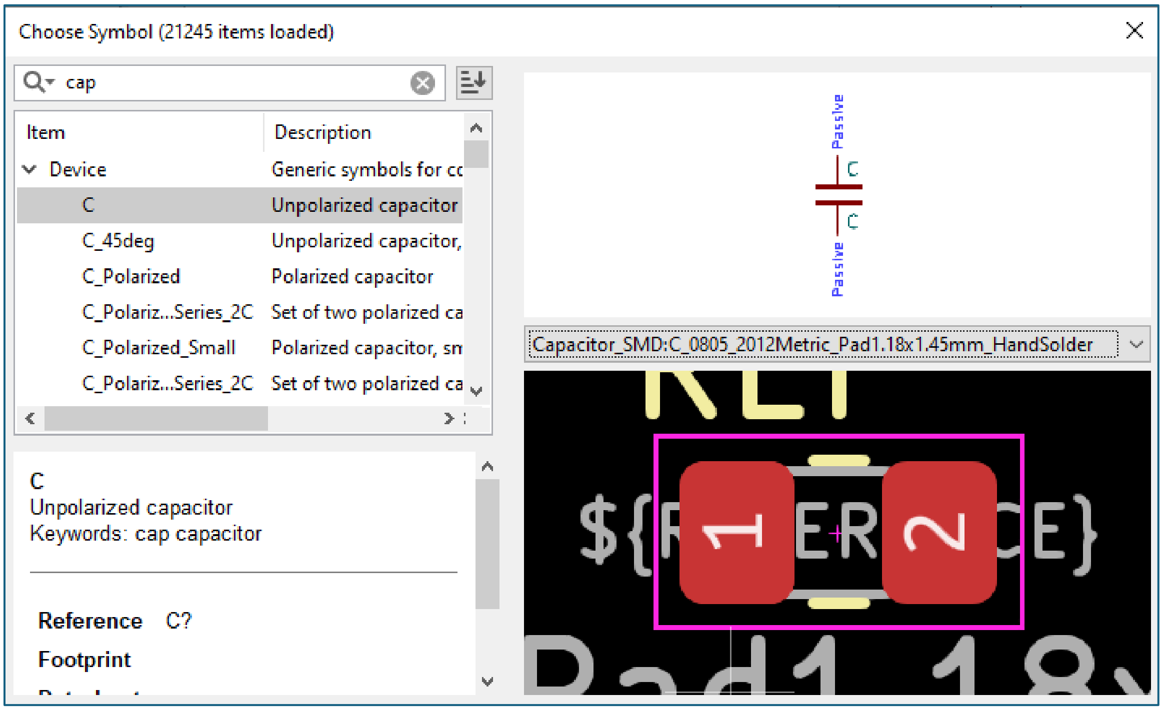


PART 13. CAPACITORS

Place the following parts, refer to the schematic for placements.

Schematic Symbol: C

Footprint: Capacitor_SMD:C_0805_2012Metric_Pad1.18x1.45mm_HandSolder



Value: 0.1uF | Datasheet:

Quantity: 1

Schematic Symbol: C | **Footprint**: Capacitor_SMD:C_0805_2012Metric_Pad1.18x1.45mm_HandSolder

Value: 1uF | Datasheet:

Quantity: 3

Schematic Symbol: C | **Footprint**: Capacitor_SMD:C_0805_2012Metric_Pad1.18x1.45mm_HandSolder

Value: 22uF | Datasheet:

Quantity: 8

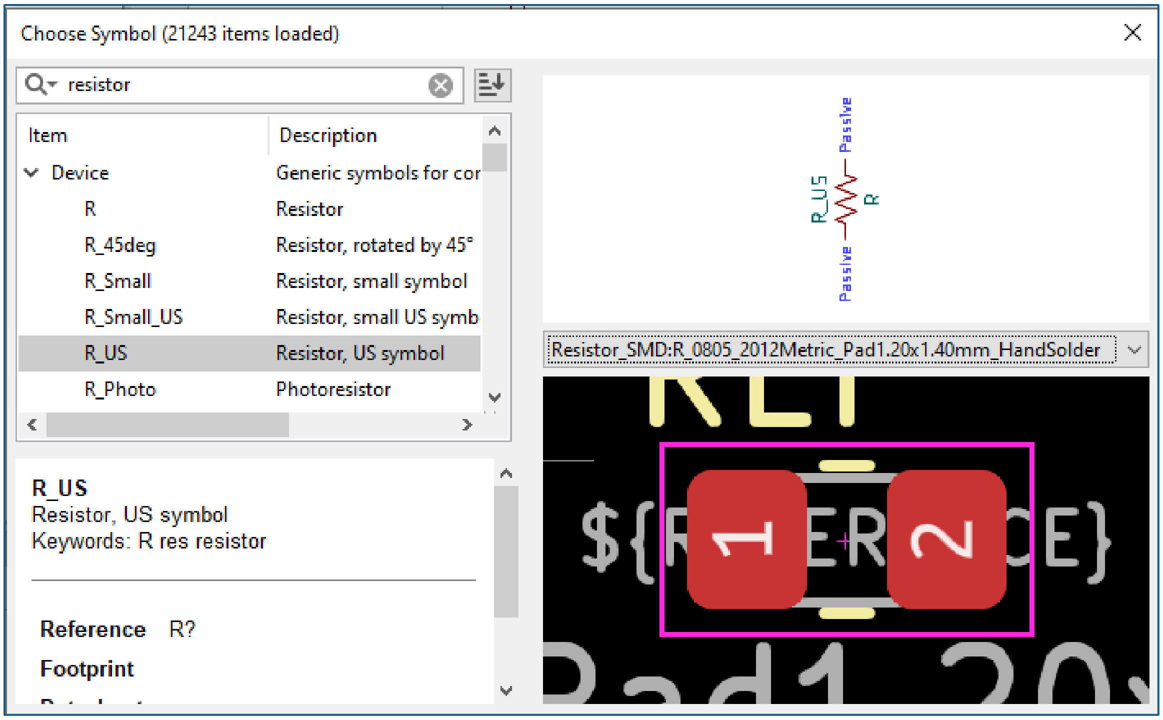
Schematic Symbol: C | **Footprint**: Capacitor_SMD:C_0805_2012Metric_Pad1.18x1.45mm_HandSolder

PART 14. RESISTORS

Place the following parts, refer to the schematic for placements.

Schematic Symbol: R_US

Footprint: Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder



Value: 330 | Datasheet: [CRCW0805330RFKEA] **DESC: SMD 560 OHM 1% 1/8W 0805**

Quantity: 4

Schematic Symbol: R_US

Footprint: Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder

Value: 1K | Datasheet: [CRCW08051K00FKEA] **DESC: SMD 1K OHM 1% 1/8W 0805**

Quantity: 4

Schematic Symbol: R_US

Footprint: Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder

Value: 5K | Datasheet: CRCW08055K00FKTA DESC: SMD 5K OHM 1% 1/8W 0805

Quantity: 4

Schematic Symbol: R_US

Footprint: Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder

Value: 160K | Datasheet: [CRCW0805160KFKEA] **DESC: RES SMD 160K OHM 1% 1/8W 0805**

Quantity: 1

Schematic Symbol: R_US

Footprint: Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder

Value: 1.5M | Datasheet: [CRCW08051M50FKEA] **DESC: SMD 1.5M OHM 1% 1/8W 0805**

Quantity: 1

Schematic Symbol: R_US

Footprint: Resistor_SMD:R_0805_2012Metric_Pad1.20x1.40mm_HandSolder

PART 15. LEDS

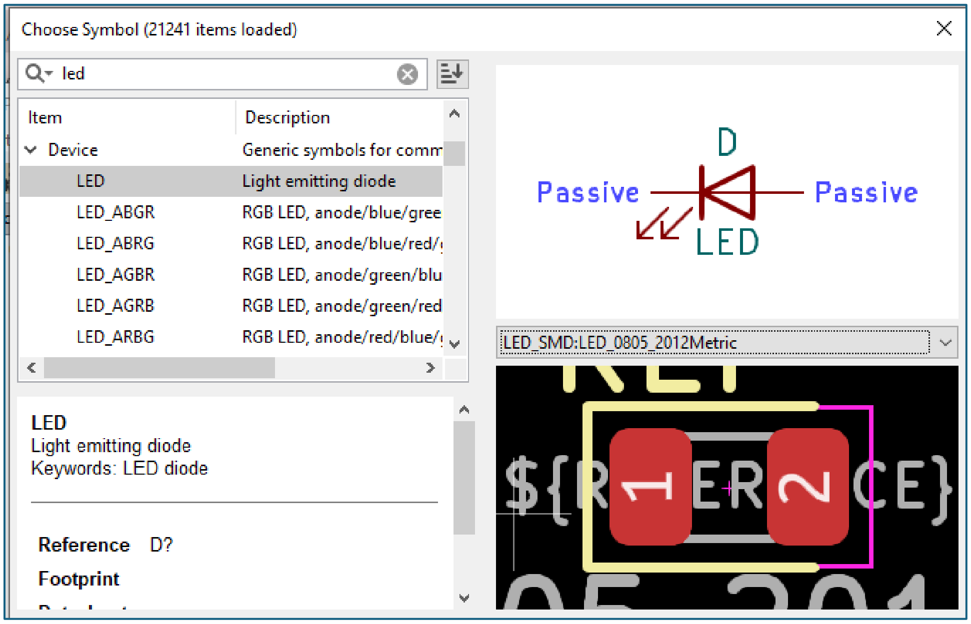
Place the following parts, refer to the schematic for placements.

Green LEDs: LTST-S220GKT **LED GREEN CLEAR SMD 0805**

Quantity: 3

Schematic Symbol: LED

Footprint: LED_SMD:LED_0805_2012Metric



PART 16. HEADERS AND TEST POINTS

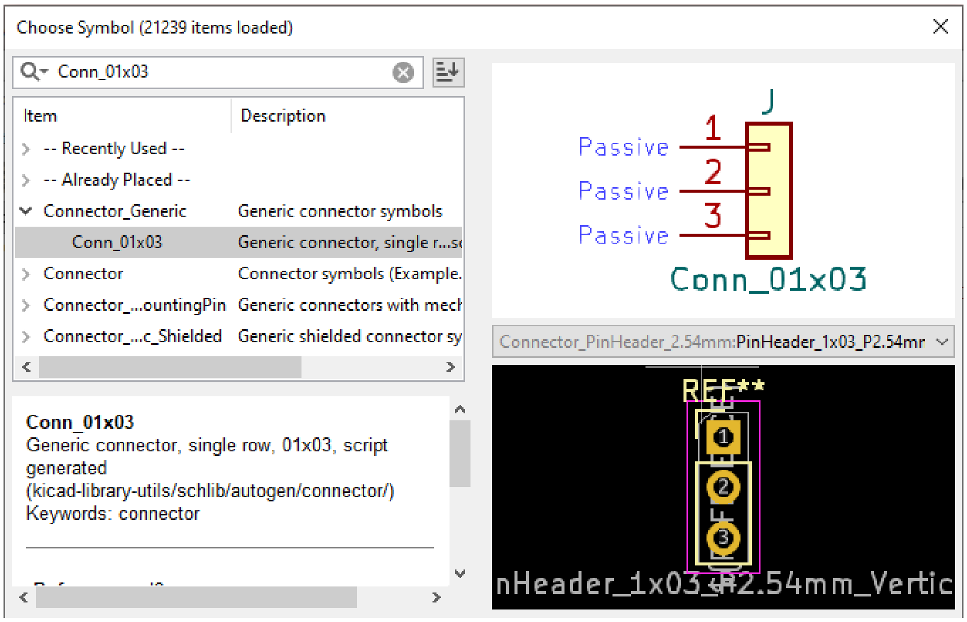
Place the following parts, refer to the schematic for placements.

Voltmeter Module Receptacle Connector

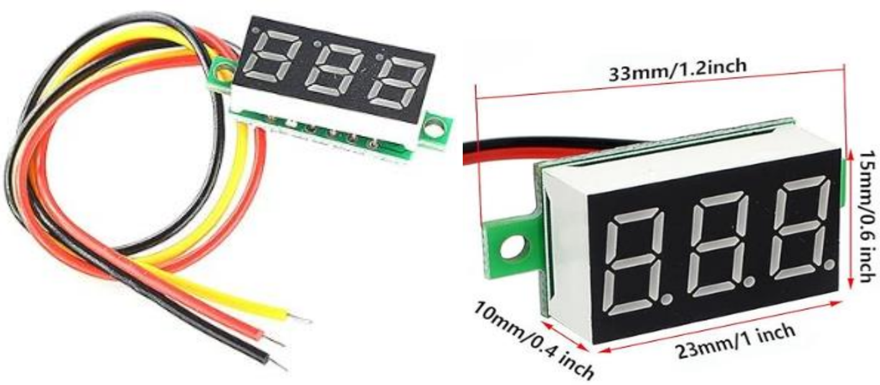
Quantity: 2

Schematic Symbol: Conn_01x03

Footprint: PinHeader_1x03_P2.54mm_Vertical



NOTES: These board pins are where the meters are soldered too. PART: Amazon/AliExpress: 3-Wire 0.28 Inch LED Panel



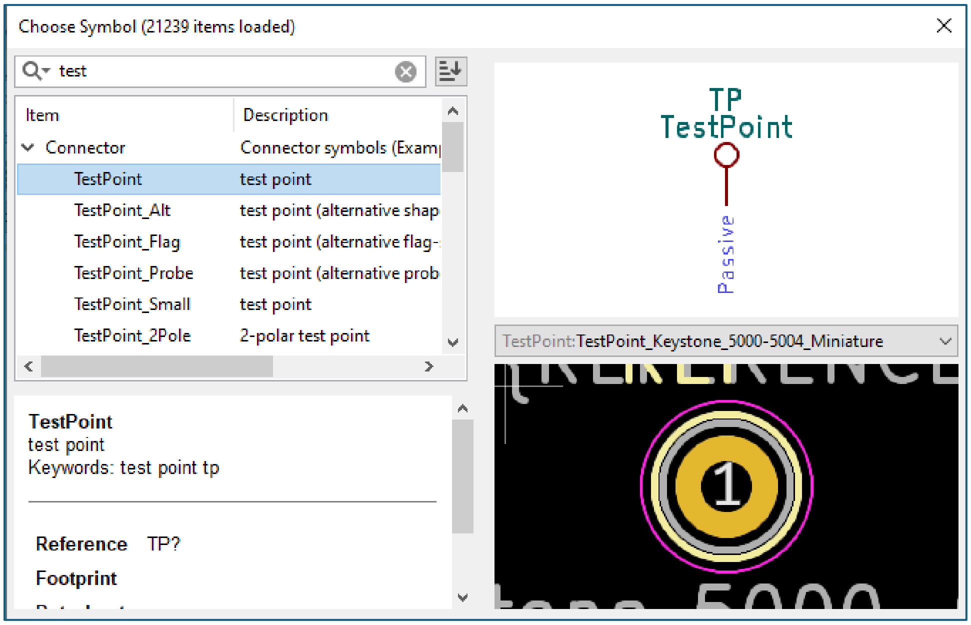
Test Points

Place the following parts, refer to the schematic for placements.

Quantity: 5

Schematic Symbol: TestPoint

Footprint: TestPoint:TestPoint_Keystone_5000-5004_Miniature



ADD WIRES, POWER SYMBOLS, AND NET LABELS

Start wiring your schematic and use power symbols when needed or preferred. Wires directly connect electrical pins, while power symbols act as connection points, like wires, but also assign net names based on their labels. Using power symbols can simplify the schematic, making it easier to read and helping with net identification during the PCB layout phase. Choose whichever method—wires or wires and power symbols—works best for your design!

<u>I've added a few net labels as well.</u> Without them, the schematic app assigns random invisible names to the wires, like NET1, NET2, and so forth. By clearly naming the nets, it becomes easier to identify important signals and assign special PCB routing parameters, such as thicker traces (wires), during the PCB layout stage. This is simply my personal preference, based on experience, and not a requirement.

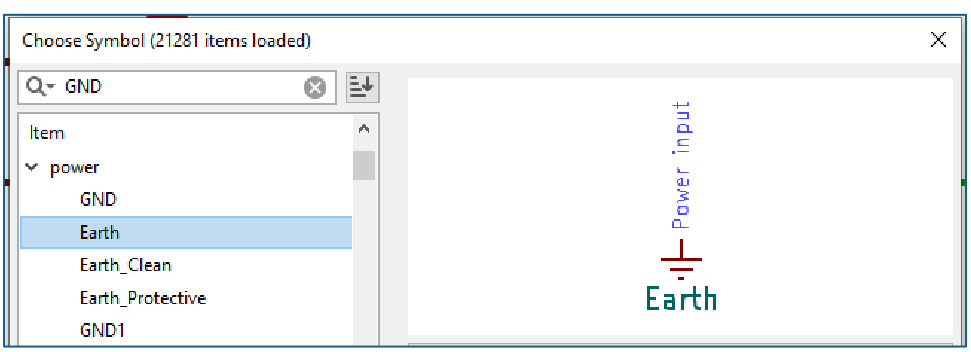
Adding and Editing Wires

To begin connecting pins with a wire, click the **Add Wire**  button on the right toolbar. Wires can also be automatically started by clicking on an unconnected symbol pin or wire end.

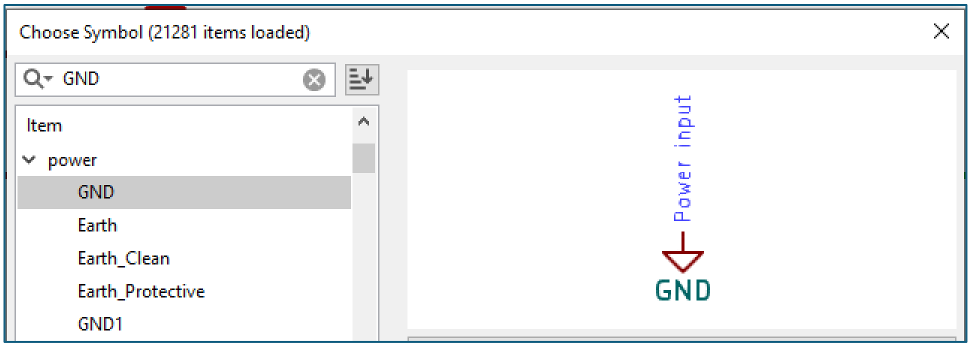
Adding Power Symbols

Click the **Add a Power**  button on the right side toolbar to insert power symbols into your schematic.

Earth Ground Symbol:



Power Ground Symbol:

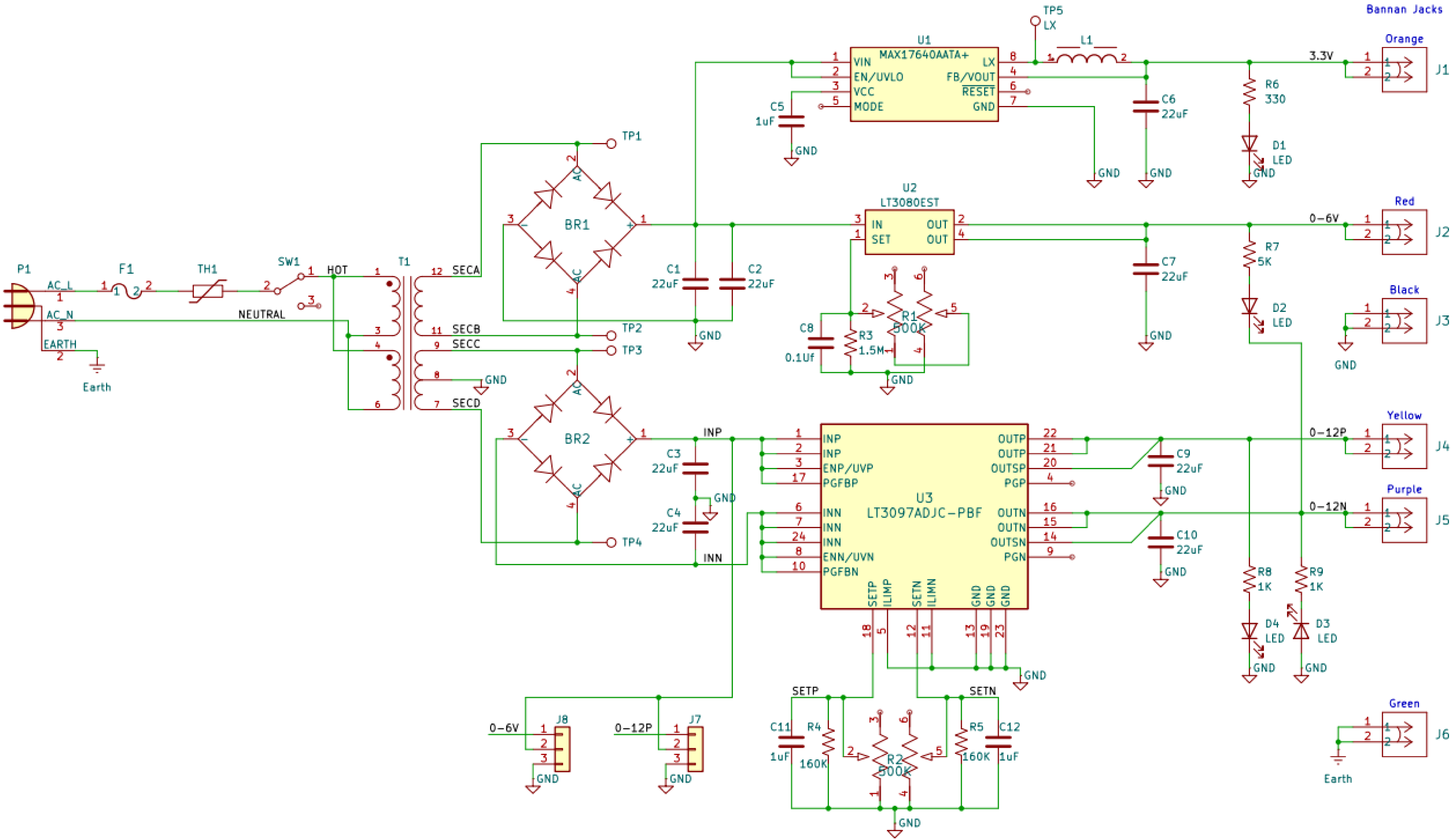


Adding Label

To add net labels, click the **Add Label** A button on the right toolbar. To assign the label to a wire, click along the path of the wire to assign.

Example SECA, SECB, SECC, SECD.

COMPLETE THE SCHEMATIC SHOWN BELOW


Bannan Jacks
TP5
LX
U1
MAX17640AATA+
VIN
EN/UVLO
VCC
MODE
LX
FB/VOUT
RESET
GND
L1
C5
1uF
GND
C6
22uF
R6
330
D1
LED
3.3V
Orange
J1
TP1
BR1
AC
U2
LT3080EST
IN
SET
OUT
OUT
0-6V
Red
J2
C1
22uF
C2
22uF
C7
22uF
R7
5K
D2
LED
C8
0.1Uf
R3
1.5M
R1
500K
Black
J3
P1
AC_L
AC_N
EARTH
Earth
F1
TH1
SW1
HOT
NEUTRAL
T1
SECA
SECB
SECC
SECD
TP2
TP3
TP4
BR2
INP
INN
C3
22uF
C4
22uF
U3
LT3097ADJC-PBF
INP
ENP/UVP
PGFBP
INN
ENN/UVN
PGFBN
OUTP
OUTSP
PGP
OUTN
OUTSN
PGN
SETP
ILIMP
SETN
ILIMN
C9
22uF
C10
22uF
0-12P
Yellow
J4
0-12N
Purple
J5
R8
1K
R9
1K
D4
LED
D3
LED
Green
J6
J8
J7
0-6V
0-12P
SETP
SETN
C11
1uF
R4
160K
R2
500K
R5
160K
C12
1uF