

Advanced LED Lantern 1.0

Instruction Manual

Eastern Voltage Research, LLC

Introduction to the Advanced Lantern 1.0 Kit

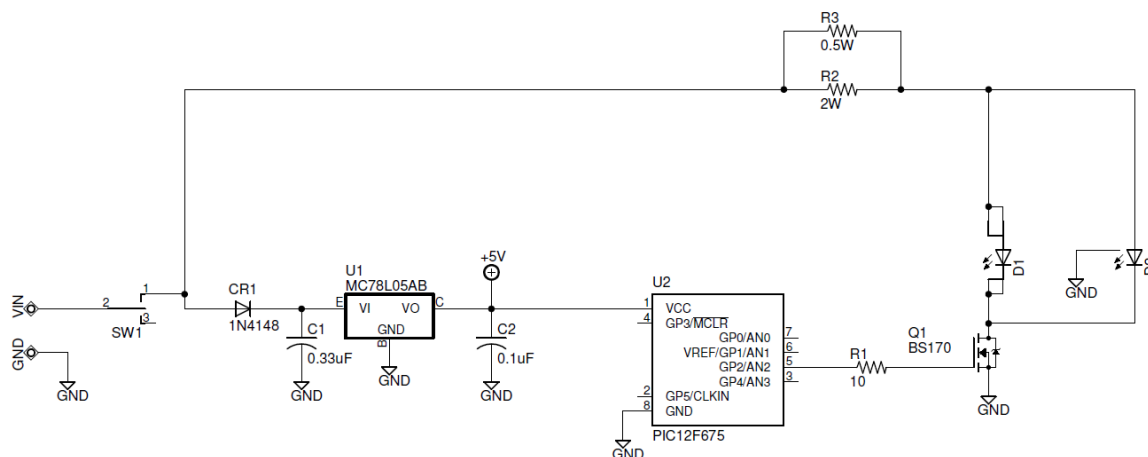
Thank you for purchasing the Advanced Lantern 1.0 Kit. This kit is an advanced microprocessor controlled high power LED lantern that provides an ultra-realistic candle flame pattern. It utilizes true random algorithms to produce a highly realistic and ultra bright flame for use any many applications. This particular kit was tailored specifically for props such as Halloween Jack O' Lanterns or for various effects in haunted houses and other "Dark Rides" and amusements.

The Advanced Latern 1.0 kit utilizes a PIC microcontroller to produce a randomly varying pulse width modulated (PWM) output signal which drives a high power CREE Piranha series amber LED or the optional ultra-high brightness 1W CREE XPE amber LED. The kit comes with a 9V battery, but can operate from any voltage source from 7V to 12VDC. An integrated ON/OFF switch is used to power the unit ON or OFF.

Notice to Beginners: If you are a first time kit builder, you may find this instruction manual easier to understand than expected. Each component in this kit has an individual check box, while a detailed description of each component is provided as well. If you follow each step in the instruction manual in order, and practice good soldering and kit building skills, the kit is next to fail-safe.



Please read this manual in its entirety before building, testing, or operating your kit!



Circuit Description

The Advanced Lantern 1.0 utilizes a PIC12F675 microcontroller, U2, which produces a randomized output pulse width modulation (PWM) signal which is used to drive a high current MOSFET transistor, Q1, to power a CREE Piranha series LED, D1 or the optional high power 1W CREE XPE LED. The output PWM signal of the PIC12F675 drives the gate of MOSFET transistor, Q1, which when turned ON, allows current to conduct through either LED, D1 or D2, and causes it to illuminate. U1 is a 5V linear voltage regulator which provides a regulated 5VDC for the PIC12F683 microcontroller.

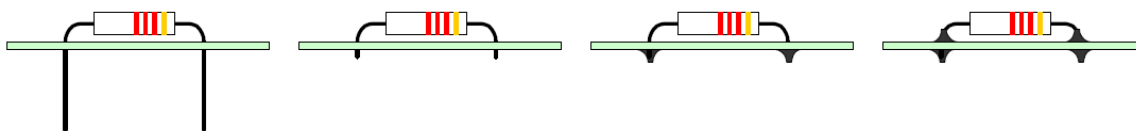
A slide switch allows the user to turn the unit ON or OFF.

Kit Building Tips

A good soldering technique is key! Let your soldering iron tip gently heat both the wires and pads simultaneously. Apply solder to the wire and the pad when the pad is hot enough to melt the solder. The finished joint should appear like a small shiny drop of water on paper, somewhat soaked in. If the pads have not heated up sufficiently, melted solder (heated only by the soldering iron itself) will form a cold solder joint and will not conduct properly. These cold joints appear as dull beads of solder, and can be easily fixed by applying additional heat to the pad and wire. All components, unless otherwise noted, should be mounted on the top side of the board. This is the side with the silkscreen printing.

When installing components, the component is placed flat to the board and the leads are bent on the backside of the board to prevent the part from falling out before soldering. The part is then soldered securely to the board, and the remaining lead length is clipped off. It is also extremely important to place the components as close to the board as possible. This is necessary for proper operation over the wide frequency range of the various kits we provide. Also be sure that component lead lengths are always as short as possible. This will avoid adding any stray capacitances or inductances that can be detrimental to circuit operation.

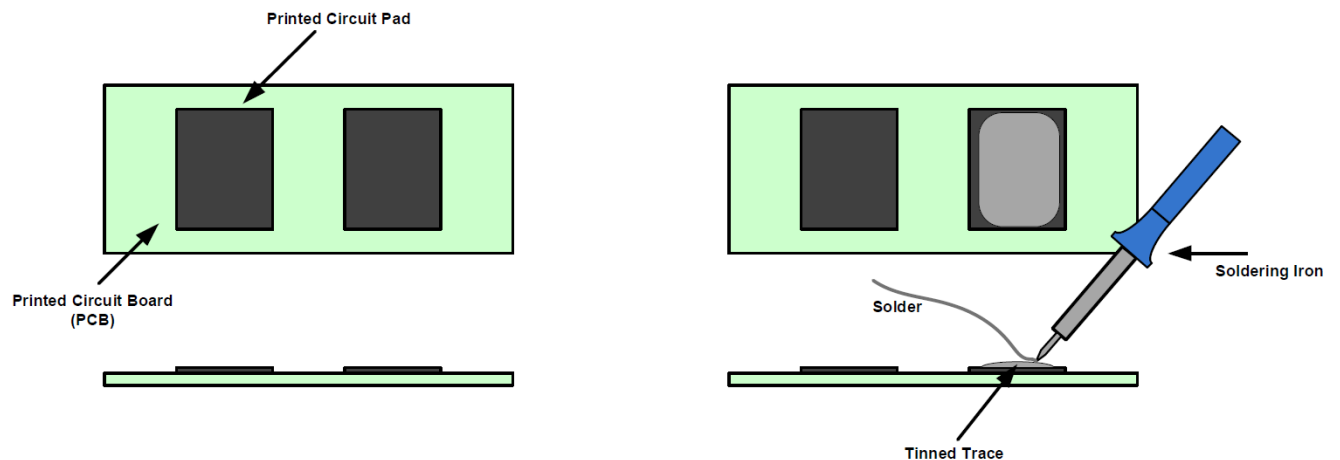
An alternative approach (which is actually the one I use) is to install the component into the board and then apply a piece of masking tape on the topside to hold the component in place temporarily. The leads on the backside of the board are then trimmed leaving about 0.10" lead protruding through the backside of the board, and then soldered from the backside. You can then remove the masking tape, and finally apply a small amount of solder on the top to complete the joint on both sides. This is shown in the figure below.



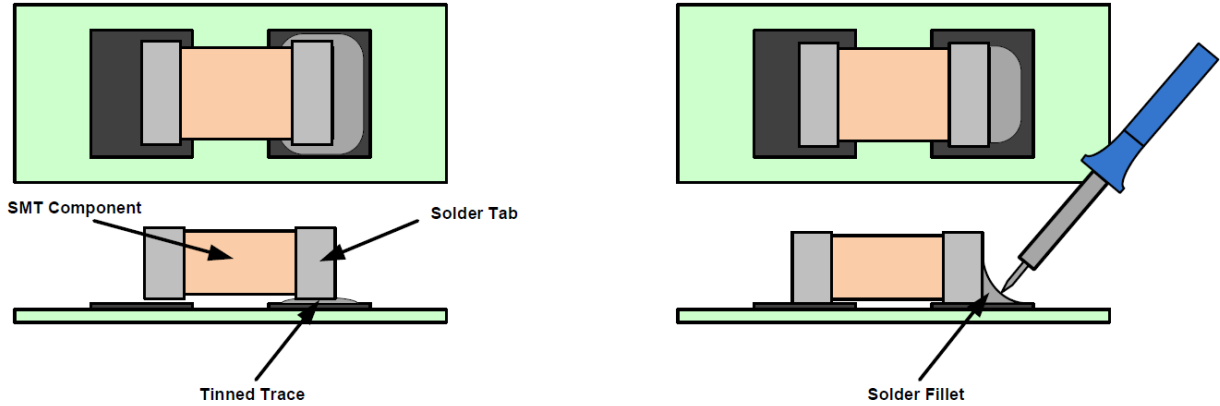
Surface Mount (SMT) Component Soldering Instructions

One of the first things you'll notice with your electronics kit is that many of the included components are surface mount components. These components do not have conventional leads, as is the case with thru-hole components, and instead solder directly to pads located either on the top or bottom of the PCB board.

One of the first things to remember when soldering surface mount (SMT) components to the board is that patience is a must! The first step when soldering a SMT component to the board, after properly identifying both the component and the location where it will be installed on the PCB board, is to slightly "tin" one of the pads on the PCB board that will connect to the component. This is accomplished by simply applying a very small amount of solder directly to the pad with the soldering iron as shown below.



The next step is to pick up and hold the component in place on its tinned pad using tweezers. While holding the component in place with tweezers, briefly re-heat the solder with the soldering iron so that it flows onto the component solder tab and forms a nicely shaped solder fillet. For the remaining solder tabs on the same component, briefly heat up the component tab using the soldering iron and apply a small amount of solder directly to the pad, again creating a nicely shaped solder fillet. It is important to note that when reheating the solder, the soldering iron tip should contact the solder tab of the body of the component and not the solder directly. This will allow the solder to flow as efficiently as possible and form a proper solder fillet.



At first, surface mount soldering may seem a bit difficult, but its actually much easier than thru-hole soldering once you get the hang of it. Good luck and take your time!

Advanced Flame 1.0 Kit Parts List

RESISTORS

- 1 10 ohm Resistor (brown-black-black), R1
- 1 68 ohm Resistor, 2W (blue-grey-black), R2 (supplied with HP kit)
- 1 100 ohm Resistor, 1/2W (brown-black-brown), R3 (supplied with LP kit)

CAPACITORS

- 1 0.33uF Capacitor, C1
- 1 0.1uF Capacitor, C2

DIODES

- 1 1N4148 Diode, CR1
- 1 LED, 4-DIP, D1 (supplied with LP kit)
- 1 LED, SMT, High Power CREE, D2 (supplied with HP kit)

SEMICONDUCTORS

- 1 MC78L05AB Voltage Regulator, U1
- 1 PIC12F675 Microcontroller, U2
- 1 BS170 MOSFET, Q1

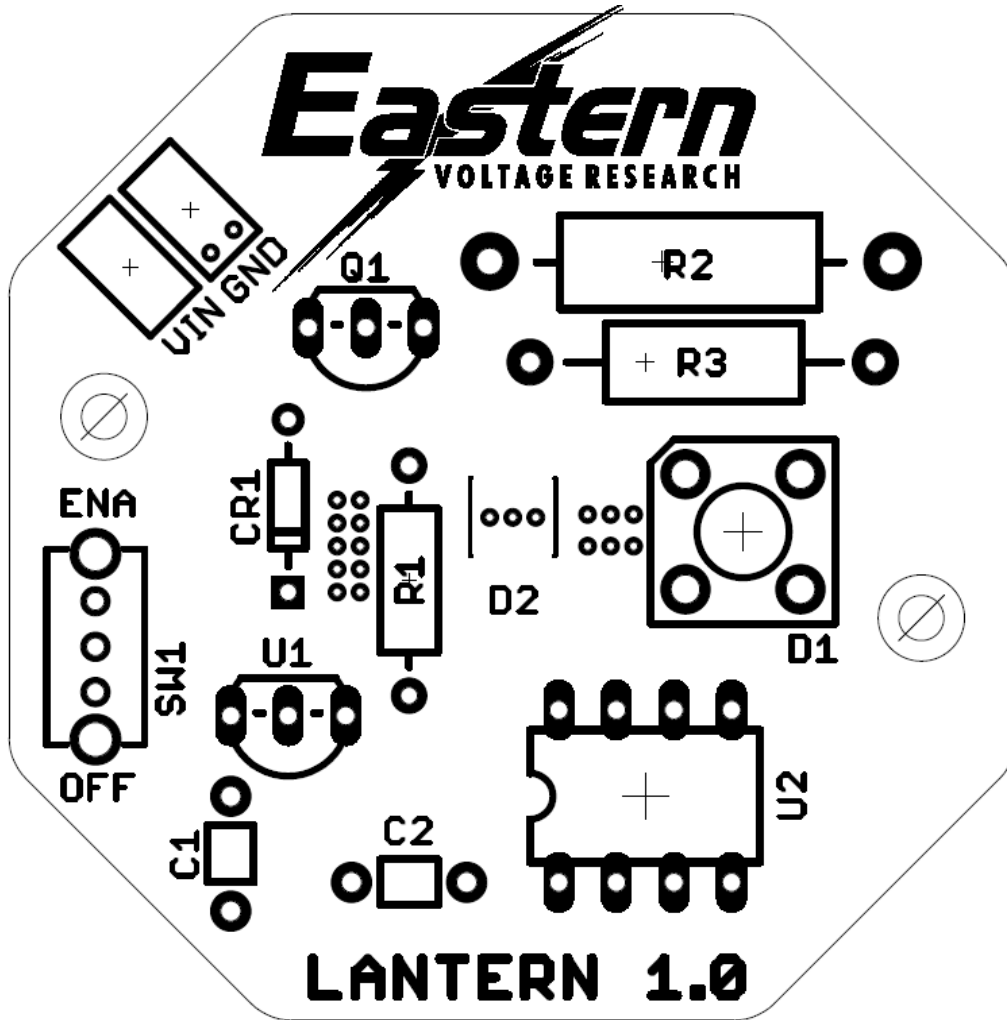
MISCELLANEOUS

- 1 Switch, SW1
- 1 9V Battery Connector
- 1 DIP Socket, 8-Pin
- 1 Printed Circuit Board (PCB), Lantern 1.0

REQUIRED, NOT SUPPLIED

- 1 9V Battery or 9-12V DC Power Supply

Advanced Lantern 1.0 - Component Layout Diagram



KIT Building Instructions

Now we will begin building the kit. There are just a few more important things to know before we install the first components.

For each component, the word “install” always means the following:

1. Pick the correct value to start with.
2. Insert the component into the correct printed circuit board (PCB) location.
3. Orient the component correctly – especially when there is a right and a wrong way to solder it in. (i.e. Electrolytic capacitors, diodes, ICs, transistors, etc...)
4. Solder all connections unless directed otherwise. Ensure enough heat is used to allow solder to flow for clean, shiny, and completed connections.

Also, please be sure to take us seriously when we say that good soldering is the key to the proper operation of your circuit!

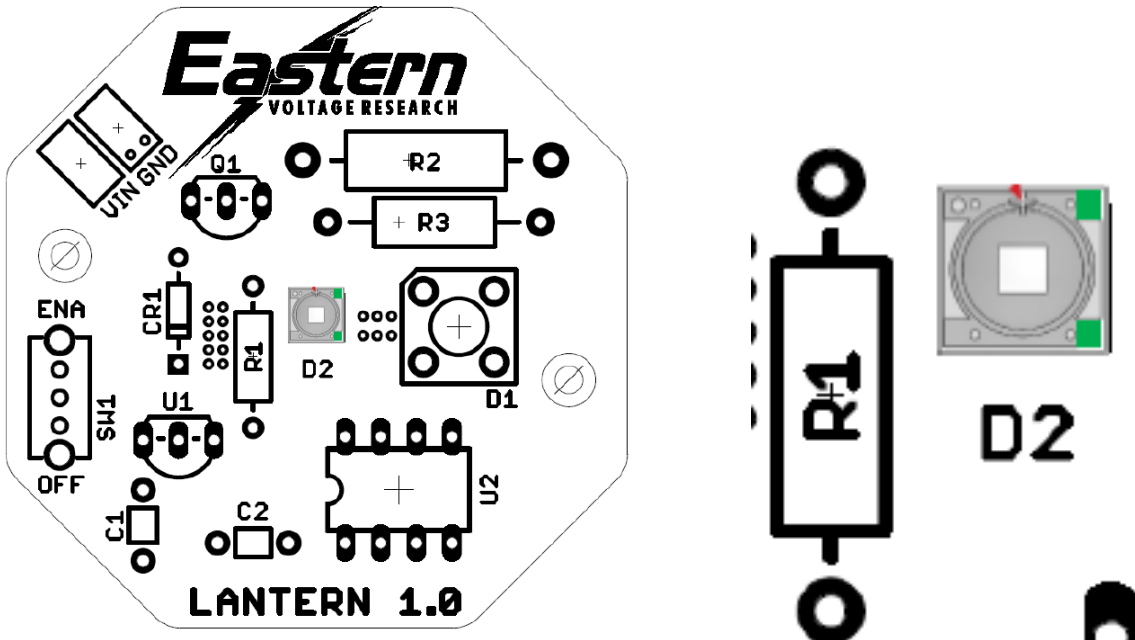
- Use a 25W soldering pencil with a clean, sharp tip. **DO NOT USE** a high power soldering gun such as those trigger activated units.
- Use only rosin core solder intended for electronics use
- Ensure your work area is clean, and has plenty of bright lighting
- Build your kit in stages, taking breaks to check your work. Be sure to clean the board periodically with a brush or compressed air to remove any excess wire cuttings, etc...

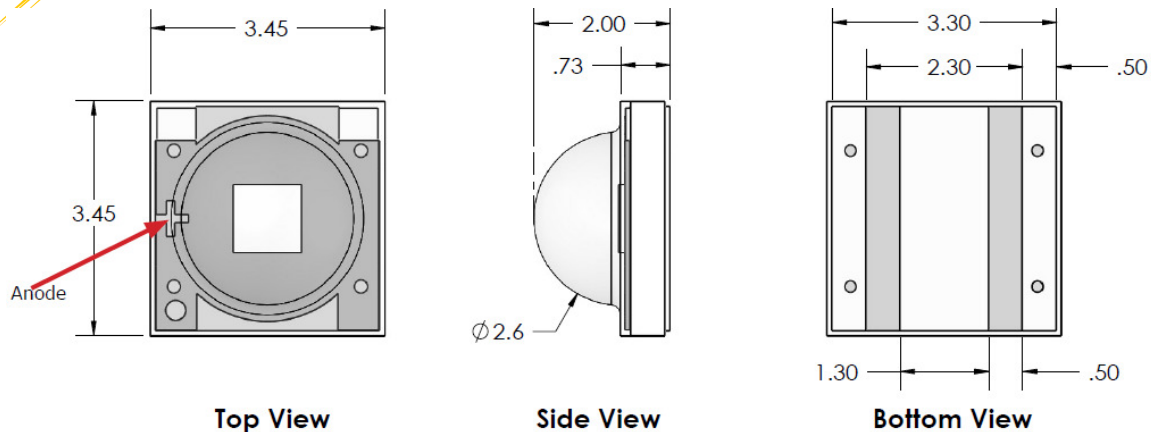
Okay, so lets begin!



DO NOT INSTALL more than one LED in this kit! Only use either the D1 or D2 LED.

- 1. Install D1, High Power SMT CREE LED. **(HP KIT ONLY)**
 Using the diagram below, install the LED with the “+” mark facing the top of the board as shown below. The easiest way to solder these components is to first apply a generous amount of flux over both of the surface mount pads of the component. Next use tweezers to place the component on the pad, and touch the soldering iron to a small bit of solder. Touch the soldering iron to one pad and the solder should flow to the component and surface mount pad. Next, touch the soldering iron to the other pad (not component) and touch a little bit of solder on the pad and it should flow on the component. Be sure not to overheat the LED or you will damage it!





- 2. Install R1, 10 ohm resistor (brown-black-black)
- 3A. Install R2, 68 ohm, 2W resistor (blue-gray-black) **(HP KIT ONLY)**
- 3B. Install R3, 100 ohm 1/2W resistor (brown-black-brown) **(LP KIT ONLY)**
- 4. Install CR1, 1N4148 diode. The cathode band on the diode must match that shown on the silkscreen.
- 5. Install C1, 0.33uF capacitor (marking BC334 or 334 or similar)
- 6. Install C2, 0.1uF capacitor (marking BC104 or 104 or similar)
- 7. Install U1, MC78L05AB voltage regulator (marked MC78L05AB) This IC needs to be orientated properly. Please insert U1 into the board with the flat edge of the IC orientated according to the silkscreen layout drawing.
- 8. Install an 8-pin DIP socket into the U2 location. Note that one end of the DIP socket is marked by a notch; this end **MUST** be orientated as shown on the PCB layout. **DO NOT INSTALL PIC12F683 Microcontroller at this time!**
- 9. Install Q1, BS170 MOSFET transistor (marked BS170) This transistor needs to be orientated properly. Please insert U1 into the board with the flat edge of the transistor orientated according to the silkscreen layout drawing.
- 10. Install D2, LED. The cathode end of the package is designated with a notch. Ensure that the notch lines up with the notch according to the silkscreen on the layout drawing. **(LP KIT ONLY)**
- 11. Install switch, SW1.

- ❑ 12. Install the 9V battery connector to the PCB board. The red wire connects to the terminal pad labeled VIN on the PCB while the black wire connects to the terminal pad labeled GND. Be sure to pass the wires through the hole in the board first prior to soldering the connector to the pads. This hole will provide strain relief for the battery wires and help prevent them from ripping off the board during handling.

Congratulations! You have just completed your Advanced Lantern 1.0 kit. Please take a few moments to look over the board and ensure that all the components are installed properly with the correct orientation. Since some of the parts may be unfamiliar to you, you may want to be extra sure that they have been inserted correctly. After you are sure that everything seems to be properly installed, move on to the set-up and testing section.

Set-up and Testing



DO NOT INSTALL more than one LED in this kit! Only use either the D1 or D2 LED.



Be sure not to stare into the LED. The LEDs are extremely bright and may cause permanent damage to the eye during prolonged exposure.

Okay, so lets begin!

- ❑ 1. Install a 9V battery into the battery connector, or connect a 7-12VDC power supply to the proper terminals ensuring proper polarity is observed.
- ❑ 2. Using a multimeter, verify that there is 5VDC at pin 1, of U2. This will ensure that the onboard 5V regulator, U1, is connected and working properly. If the voltage is not 5V, you will need to determine why U1 is not working before continuing on. Be sure the switch, SW1, is in the “ON” position.

- 3. Disconnect the 9V battery or 7-12VDC power supply.
- 4. Install the programmed PIC12F675 microcontroller into the U2 location. Note that one end of the PIC12F675 is marked by a notch; this end **MUST** be oriented as shown on the PCB layout.
- 5. Reconnect the 9V battery or 7-12VDC power source.
- 6. The unit should begin operating and displaying a super bright LED flame!

Congratulations! Your Digital Lantern 1.0 is now completed and operational.

Troubleshooting

PROBLEM: The LED does not illuminate when I apply power.

SOLUTION: Verify that the LED D1 or D2, U1, U2, CR1, and Q1 are installed properly. Also be sure the switch, SW1, is in the “ON” position.

Conclusion

We sincerely hope that you have enjoyed the construction of this Eastern Voltage Research Kit. As always, we have tried to write this instruction manual in the easiest, most “user friendly” format that is possible. As our customers, we value your opinions, comments, and additions that you would like to see in future publications. Please submit comments or ideas to:

Eastern Voltage Research, LLC

Technical Support
support@easternvoltage.com

Thanks again from the people here at Eastern Voltage Research.

Terms and Conditions of Sale

Before opening or assembling your kit, please read and review the latest Terms and Conditions of Sale on our website at the following link:

<http://www.easternvoltage.com/terms.html>