Introduction to the Digital Flame 1.0 Kit

Thank you for purchasing the Digital Flame 1.0 Kit. This kit is definitely a favorite of mine. I use the ones I’ve built all the time, especially when camping or when I want a little ambience around the house. Its also fantastic to use for Halloween Jack O’ Lanterns!

In the past several years, I’ve purchased many of those “inexpensive” LED candle lights and have been disappointed in the performance of most of them. Most of them were either not bright enough, or simple had a repeating pattern of illumination that was readily discernible to the eye or just simply did not look realistic at all. So that is why I decided to create my own LED candle.

The Digital Flame 1.0 kit utilizes a PIC microcontroller to produce a randomly varying pulse width modulated (PWM) output signal which drives a bank of four (4) high power yellow / amber LEDs. This kit is similar to our Digital Candle 1.0 kit except that we really increased the LED output and used high power LEDs with the Digital Flame 1.0. This digital candle is now SUPER BRIGHT! We also provided a standard serial programming interface on the board so that you can experiment with your own microcontroller code!

This kit is definitely one of my favorites and I hope you enjoy it too!
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 Digital Flame 1.0 utilizes a PICF675 microcontroller, U1, which produces a randomized output pulse width modulation (PWM) signal which is used to drive a high current MOSFET transistor, Q1, to power an array of four (4) high power LEDs, D1, D2, D3, and D4. The output PWM signal of the PIC12F675 drives the gate of MOSFET transistor, Q1, which when turned ON, allows current to conduct through the LEDs (D1-D4) and causes them to illuminate.

There is also a light activated switch circuit which will only turn on the Digital Flame 1.0 when it becomes dark. This is perfect for use outside with Halloween Jack O' Lanterns or other applications. Photosensor, R7, is used to sense ambient light and to turn on MOSFET transistor, Q2, which acts as a switch. When ambient light becomes low enough (dark), Q2 turns on and therefore allows the Digital Flame 1.0 to start working. Potentiometer R8 allows the user to adjust the light level required to turn on the Digital Flame 1.0 circuit. The light activation can also be completely disabled as well if desired.

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 the 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](http://www.EasternVoltageResearch.com/DigitalCandle1.0)
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 it's actually much easier than thru-hole soldering once you get the hang of it. Good luck and take your time!
## Digital Flame 1.0 Kit Parts List

### RESISTORS

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>47k Resistor, 0805, R1</td>
</tr>
<tr>
<td>4</td>
<td>16 ohm Resistor, 1206, R2,R3,R4,R5</td>
</tr>
<tr>
<td>2</td>
<td>10 ohm Resistor, 0805, R6, JMP1</td>
</tr>
<tr>
<td>1</td>
<td>Potentiometer, 1Meg, 1T, R8</td>
</tr>
<tr>
<td>1</td>
<td>Photocell, R7</td>
</tr>
</tbody>
</table>

### CAPACITORS

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.1uF Capacitor, 1206, C1</td>
</tr>
</tbody>
</table>

### DIODES

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>LED, Yellow, PLCC-4, D1,D2,D3,D4</td>
</tr>
</tbody>
</table>

### SEMICONDUCTORS

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>DMG6968 MOSFET, SOT-23, Q1</td>
</tr>
<tr>
<td>1</td>
<td>Si2305DS MOSFET, SOT-23, Q2</td>
</tr>
<tr>
<td>1</td>
<td>PIC Microcontroller, U1</td>
</tr>
</tbody>
</table>

### MISCELLANEOUS

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Switch, Slide, SW1</td>
</tr>
<tr>
<td>4</td>
<td>AAA Battery Clips, BAT1,BAT2</td>
</tr>
<tr>
<td>1</td>
<td>PCB Board, Digital Flame 1.0</td>
</tr>
<tr>
<td>1</td>
<td>Schematic, Digital Flame 1.0</td>
</tr>
</tbody>
</table>

### REQUIRED, NOT SUPPLIED

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>AAA Batteries, BAT1,BAT2</td>
</tr>
</tbody>
</table>
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 let's begin!

☐ 1. Install R1, 47k, 0805 resistor (marking 473 or similar)
☐ 2. Install R2, 16 ohm, 1206 resistor (marking 16R or similar)
☐ 3. Install R3, 16 ohm, 1206 resistor (marking 16R or similar)
☐ 4. Install R4, 16 ohm, 1206 resistor (marking 16R or similar)
☐ 5. Install R5, 16 ohm, 1206 resistor (marking 16R or similar)
☐ 6. Install R6, 10 ohm, 0805 resistor (marking 10R or similar)
☐ 7. Install C1, 0.1uF 1206 capacitor (no marking). This is a rectangular brown component.
8. Install D1, D2, D3, and D4, LEDs. There is a notch in one corner of the LED. Each LED should be oriented as shown in the figure below for each of the (4) locations. Be sure each LED is centered on the (4) pads. Failure to orient the LED properly will result in the board not working.

9. Install U1, PIC12F675 Microcontroller. Note that one end of the IC is marked by a dot, notch, or band; this end MUST be oriented as shown on the PCB layout.

10. Install Q1, DMG6968 (marking ) MOSFET. This component has three leads and can only be installed one way on the board so no confusion there.

11. Install Q2, Si2305DS (marking ) MOSFET. This component has three leads and can only be installed one way on the board so no confusion there.

12. Install R7, Photocell. This device is a thru-hole component and has a zig-zagging line on the top of it. The photocell is not polarized and can be oriented in any direction.

13. Install R8, 1Meg Potentiometer

14. Install SW1, switch. Note that there is no incorrect way to install this switch. The switch may be inserted in any orientation.

15. Install the (4) AAA battery clips to the underside of the board.

Congratulations! You have just completed your Digital Flame 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.
PIC Microcontroller Code Modification

One of the cool options of this kit is that we have made it easy for you to experiment with your own PIC microcontroller code. We have supplied the kit with both an unprotected PIC microcontroller as well as provided pads so that you can connect a suitable flash programmer to the device to upload your own code. All that is required is an appropriate FLASH programmer and external 5VDC power source which can be connected to the board as shown below. A 1uF, capacitor is also required.

To connect a FLASH programmer to the LED sequencer board, simply connect the appropriate programmer to the board as shown below. Pads for each of the necessary signals are provided on the underside of the Digital Flame board.

1. Install 1uF, 1206 (or similar) ceramic capacitor to the 1206 pads as shown in the figure below. Any value capacitor from 1uF to 10uF (rated for at least 10VDC) will work. The capacitor can be a ceramic or electrolytic type. If using a electrolytic capacitor, the top pad on the capacitor pads is the positive pad.

2. Connect a 5VDC power source to the pads as shown below.

3. Connect the FLASH programmer to the pads as shown below.

Note: When programming the Digital Flame board, please remove all AAA batteries and connect your own 5VDC power source. The board must be powered via 5VDC for the programmer to work correctly.
Set-up and Testing

Okay, so let's begin!

1. Install (2) AAA batteries into the battery clips. Make sure the batteries are installed with the correct polarity.

2. Turn the switch to the ON position when you are in a dark environment. If everything was built and installed correctly, the LEDs should be illuminated and randomly flashing simulating the flaming characteristics of a candle.

3. Potentiometer R8 will adjust the sensitivity of the photocell circuit. If you are in a dark room and the unit is not working, simply adjust R8 until the unit illuminates. You may have to fine tune depending on the ambient light you wish the unit to trigger in.

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

Disabling the Light Detection Circuit

The light detection circuit can be disabled by simply setting the R8 potentiometer to its maximum value (1Meg) and installing a zero ohm or 10 ohm (included) jumper at the JMP1 location.

External Photocell Sensor

You can also choose not to install the R7, Photocell on the board if you wish to have it externally mounted. Simply remove R7, attach flying leads to each of the photocell’s leads, and wire into the R7 location on the PCB board. A good example of this application is a Halloween Jack O’Lantern. You would simply have the photocell sensor wired to the outside of the pumpkin, and then have the leads wired back to the digital flame PCB board. Simply cut a small hole in the pumpkin, feed the sensor through it, and have the sensor exposed to the ambient light outside of the pumpkin. Works great!

Troubleshooting

PROBLEM: The LEDs do not illuminate when I turn the switch ON.
SOLUTION: Verify that the LEDs and U1 are installed correctly, the battery is installed correctly, and the battery is not completely drained.
PROBLEM: One or more LEDs is not illuminating, but others are working.
SOLUTION: Check the soldering joints of the resistors and LEDs on each string and ensure they are properly soldered with nice clean solder joints. Also, double check that the LED is installed in the correct orientation.

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@easternvoltageresearch.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.easternvoltageresearch.com/terms.html