

Ca Bling!

Pacificon 2011

Norcal Buildathon Project

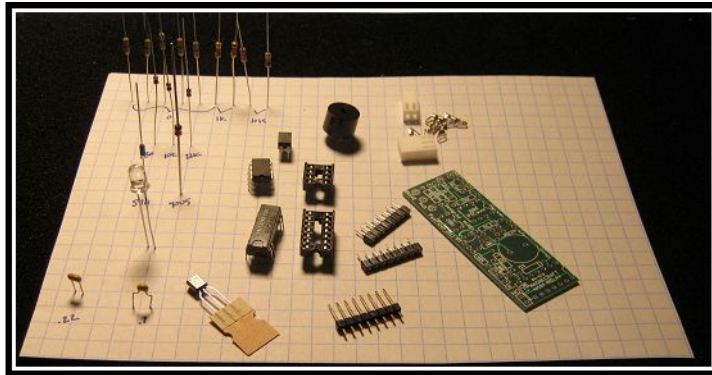
10/23/2011 ver 1.1

by **W1REX / QRPme**

www.QRPme.com



The Ca Bling! Kit is a small Picaxe micro controller development board designed by W1REX as a project board for the Norcal buildathon session as part of Pacificon 2011. The board got its name from the initial program that was developed to show off its features: microcontroller based smarts, opto-coupled input trigger, ultra-bright LED, speaker and relay. The demonstration application, the Ca Bling! program, is designed to output a short message in Morse Code when triggered by a switch. The intention was to feed the board power from the rear tail light or license plate circuit and trigger the message transmission by tapping the brake pedal. The ultra-bright LED would then send a coded message to the following vehicle. Of course, you can write any program you want to work with the resources on the circuit board.

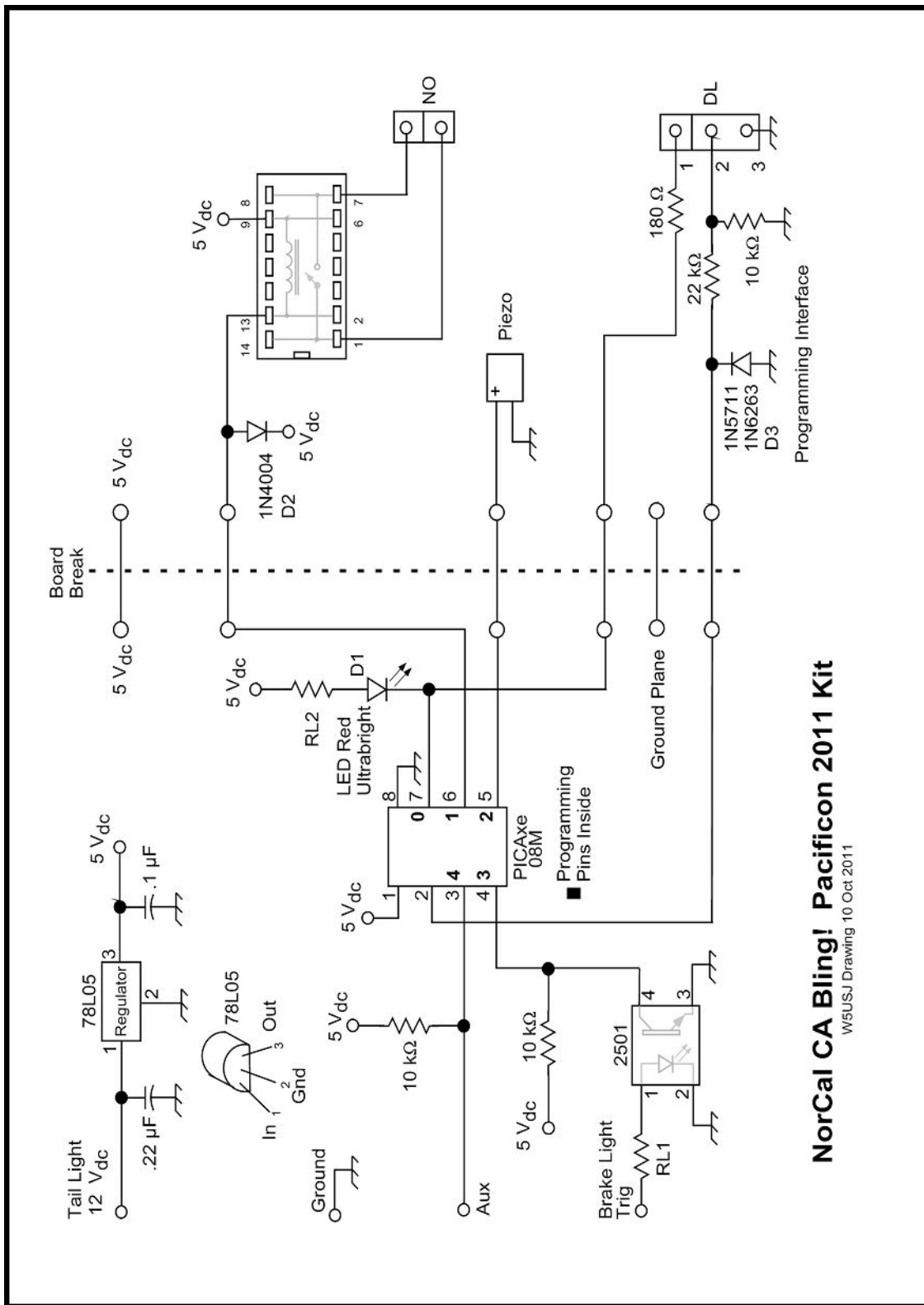


Sort out the parts for a smooth build..

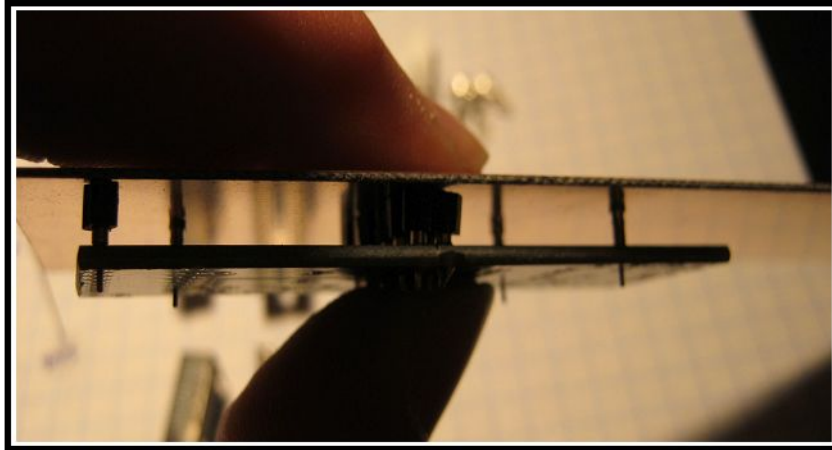
Ca.Bling! Bill of Materials

- printed circuit board
- 8 pin DIP socket
- 14 pin DIP socket
- Magnecraft W107 DIP relay
- PICAXE 08M microprocessor
- 2501 optocoupler
- LM78L05 voltage regulator
- 1N4005 diode @ D3
- 1N6263 or 1N5711 diode @ D2
- ultra-bright red LED @ D1
- piezo sounder
- 180 ohm resistor (1/8 watt)
- 1K ohm resistor (qty 2) @ RL1 & RL2
- 10K ohm resistor (qty 2)
- 10K ohm resistor (1/8 watt @ DL)
- 22K ohm resistor (1/8 watt)
- .22uf cap
- .1uf cap
- 1x6x.1" SIP Swiss pin female sockets @ snap line pads F
- 1x6x.1" SIP male pin header @ snap line pads M
- 1x8x.1" SIP Molex male header @ DL & NO
- 1x3x.1" SIP Molex female connector shell @ DL
- 1x2x.1" SIP Molex female connector shell @ NO
- 6 Molex socket terminal

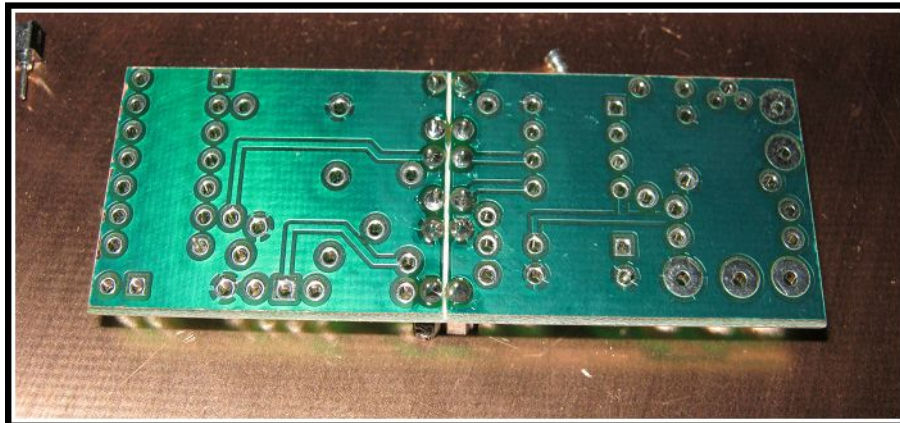
Now you should familiarize yourself with the schematic diagram of the Ca Bling! Circuitry.....



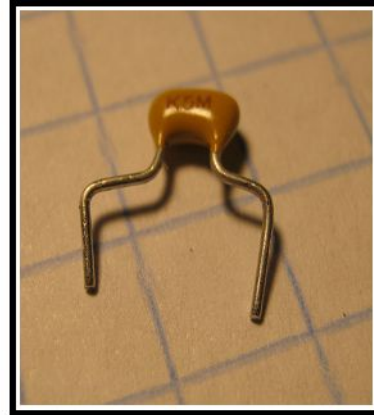
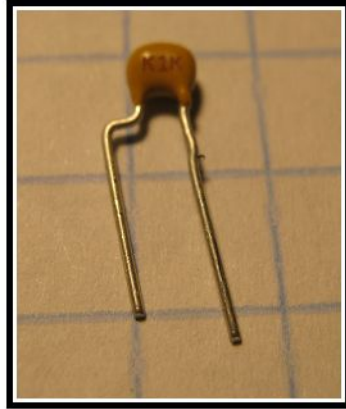
I've found that it is easiest to install all the Swiss pins first...



So I separate the strips of Swiss pins into single pins and populate the pads along the score line. I also use some of the extra pins out at the furthest pads to add stability...then place a spare piece of pcb stock over the top.

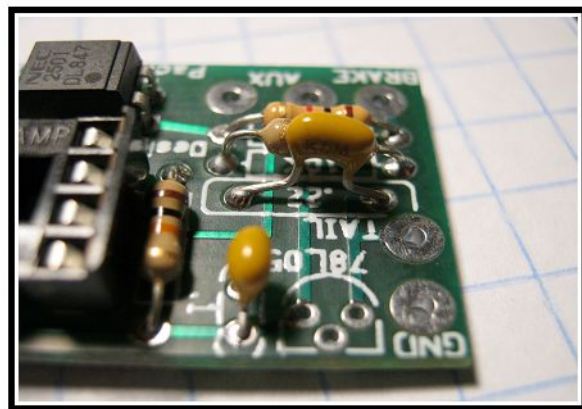
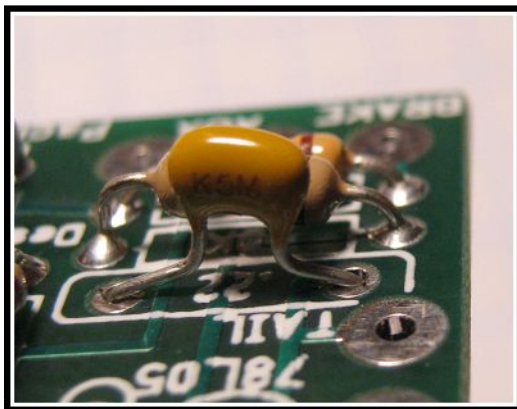


Turn the 'sandwich' over and place it on the bench. If you add a little down pressure with your finger while you solder the pins, they should all come out fairly straight and even. Just remember which pins you want to solder...and which pins you added to enable the sandwich to lay flat and even while soldering...

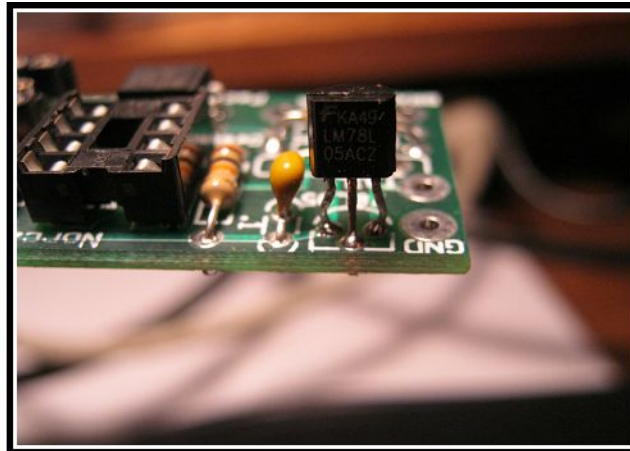


The 2 capacitors are next. You might have to re-bend the caps supplied in your kit to fit the pad spacing. The initial stock of .1uf caps had .2" spacing for .1" spaced pads so it should be re-bent like the left hand picture above. The .22uf cap has .1" spaced leads and needs to be widened for the .2" spaced pads. Bend it like the right hand picture. Install both caps.

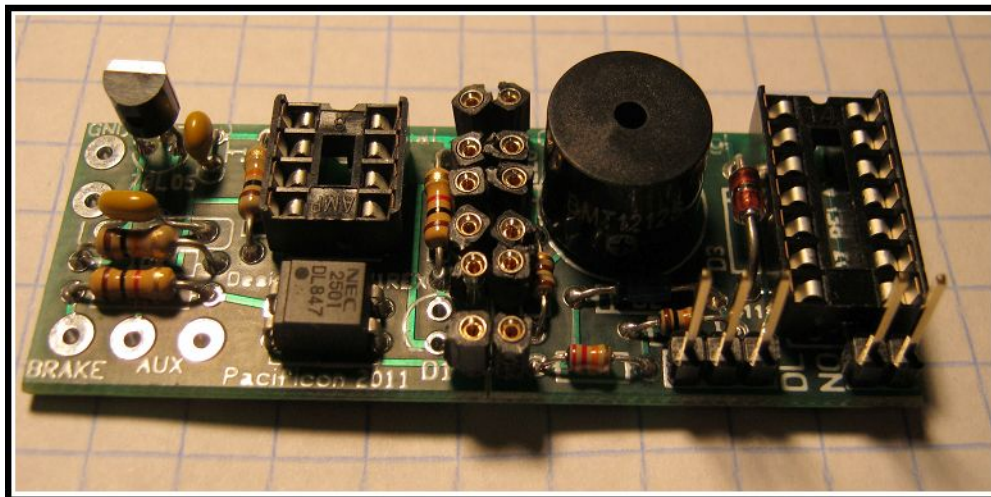
The installation of the caps should look like the following pictures.



Now solder in the 78L05 voltage regulator.



Add the small round speaker to the 'goodies' side. The speaker has a polarity so solder the speaker in with the + marked pin nearest the center of the pcb. The - pin is close to the edge of the board. You can tell the - pad on the bottom side of the pcb by its little 'star pattern' of traces connecting the pad to the ground plane. The 2 Molex style connectors (3 pin and 2 pin male headers) should now be installed. Now that most of the soldering is done, you can install the micro and relay into their respective sockets.



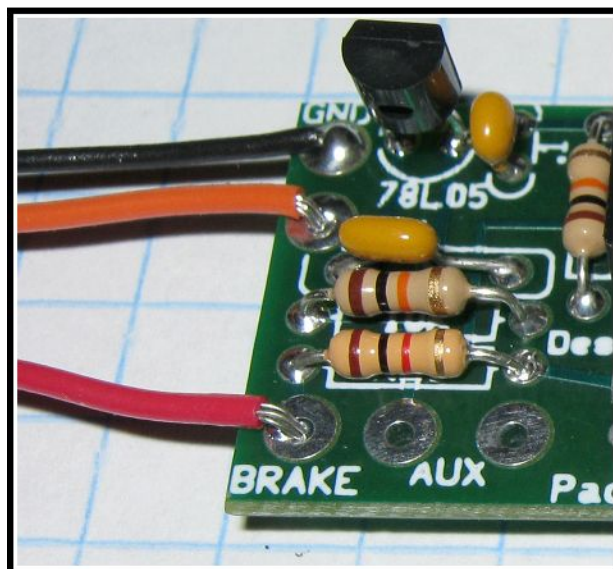
Your board should look like the picture above...

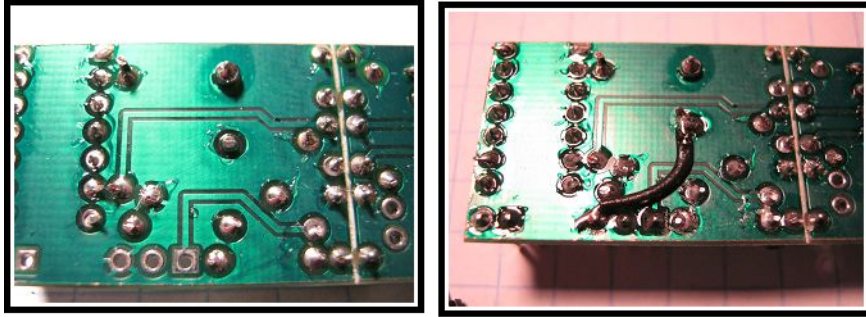
Solder in the ultra-bright LED to the pads marked D1. Note the polarity marked on the pad with a flat side. Install the LED with flats matched.

Leave some height on the LED to allow jumpers to be installed/uninstalled. Basic Ca Bling! Operation only requires 3 jumpers to download new programs into the micro but you might as well connect all 6 jumpers for testing purposes.



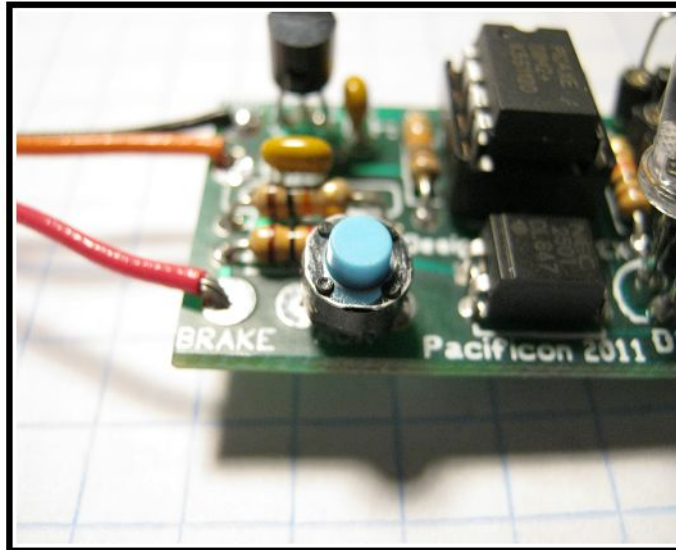
The Ca Bling! Board requires 3 wires to operate. Ground comes into the board at GND. The +12 volt supply comes into the board at TAIL while the Ca Bling! trigger (switch) signal enters through BRAKE. Solder 3 wires to those pads.





Unfortunately, the ground plane did not flow to well on the 'goodies' side of the board. A portion of the ground plane containing the speaker ground is isolated from the ground on the other half. Solder a short piece of hookup wire between the - speaker lead and the ground pin of the downloading connector as shown in the upper right picture.

I soldered a small computer front panel switch from my junk box to the AUX terminals to serve as an alternate switch for program development.



Your Ca Bling! Kit should now be ready to apply power to and download the Ca Bling! Program. But you will need to make the downloading cable.

Here are the details for the cable:

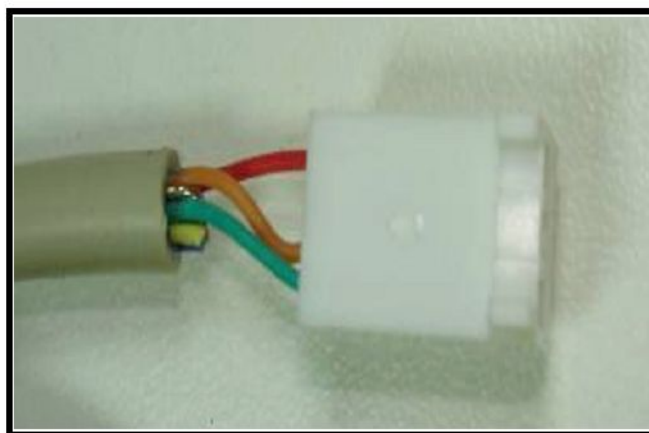
The downloading cable can be made using an 'old' DB9F serial cable like the one pictured. Below. One end is cut off and 3 Molex pins are inserted on the Txd, Rxd and Ground (2,3 and 5) pins. Complete details for making both Serial and USB downloading cables can be found at the Picaxe site:

< http://www.rev-ed.co.uk/docs/picaxe_manual3.pdf>

Use the link to the manual and go to Page 42 for cable info...



Green=ground
Red & Orange
Are Tx & Rx



Mark the ground end to keep track when attaching to your project board.
Mark the board ground too.



A homebrew download cable ready to go...

Check into the QRPme website:

www.QRPme.com

OOPS! The following picture shows the Ca Bling! Board completed and ready for some serious programming FUN....with 6 errors Can you spot them?

