Higher power, loads more features . . .

Deluxe Touchscreen eFuse

PCB assembly and calibration

We introduced our new Deluxe eFuse last month and described its hardware. Now we will get onto building it. All components mount on a single PCB which then attaches to the front panel and the whole thing fits into a Jiffy box. We'll also go over the testing and calibration procedures.

Part two: by Nicholas Vinen

This **Deluxe eFuse** can handle higher currents and voltages than our earlier and simpler eFuse design in the April issue (www.siliconchip.com.au/Article/10611).

Based around the Micromite LCD BackPack, it uses a 320x240 colour touchscreen for feedback and control. Last month we featured the full circuit and described how it works, and now we will cover its construction.

Before you start, you'll need to obtain the PCB and gather the various parts, as detailed in the parts list below. We have made a few changes in the circuit, in the light of having built a second prototype, and these have been incorporated into the final PCB design.

As shown in the circuit diagram last month (Fig.3), PNP transistor Q2 is driven directly from the output of IC2b.

This causes a problem when Q1 is switched off, ie, when the positive load is disconnected (as it is when the unit is first powered on).

That's because IC2's negative power supply is ground (0V) and when Q1 is switched off, as there is no current feedback, IC2b's output will try to go down to 0V.

That is below V+H (ie, the V+H rail is about 10V below V+ so normally at least +2V) and this will cause Q1's collector-base junction to become reverse-biased.

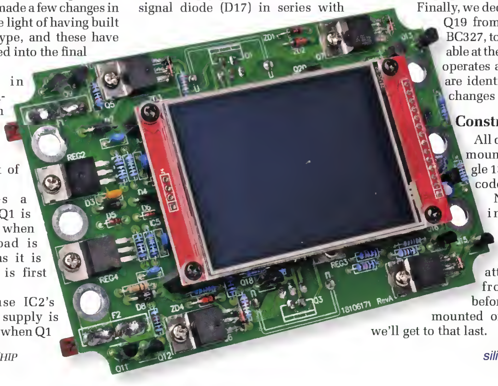
This, in turn, allows IC2b to pull down the V+H rail, increasing its own supply voltage, potentially to damaging levels.

Luckily, the solution is simple: we've simply placed a 1N4148 small signal diode (D17) in series with Q2's collector, preventing its collector-base junction from becoming reverse biased.

We're also changing the two 22Ω resistors to 15Ω, as we discovered that the SenseFET current ratio for Q1 and Q3 is 500:1, not 1000:1 as we stated in the first article.

Thus, the current through these sense resistors is twice what we had expected and so the voltages are also doubled. Reducing the resistor values brings the operation back closer our initial design parameters.

Finally, we decided to change Q19 from a BC557 to a BC327, to ensure it's reliable at the current level it operates at. The pinouts are identical so no PCB changes were required.

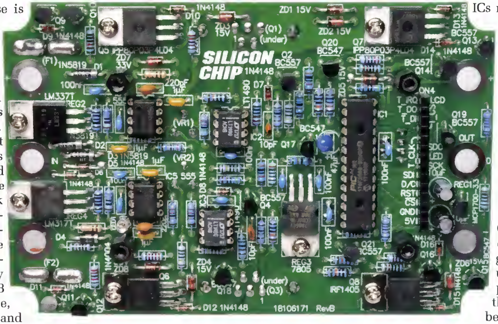


Construction

All components are mounted on the single 132 x 85mm PCB coded 18106171.

Note that the input/output binding post/banana terminals need to be attached to the front panel/lid before they can be mounted on the PCB, so we'll get to that last.

Everything else is soldered to this PCB first. Refer to the overlay diagrams, Fig.4 (top side) and Fig.5 (bottom side) as guides during assembly.



As noted last month, while this circuit is based on the Micromite LCD BackPack (originally described in February 2016), we have incorporated its circuitry on the same PCB as everything else, to reduce cost and simplify construction.

So you only have to assemble the one PCB.

Start by fitting all the resistors. It's best to check the values using a multimeter since the colour bands can be easily confused (eg, orange can look like red). The two 1Ω resistors are 0.5W types and may be 5% while all the others can be 0.5W or 0.25W 1% metal film.

Follow with the diodes and zener diodes, taking care to orientate them as shown in Fig.4.

Note that there are three different diodes types used: 1N4004 (x1), 1N4148 (x13) and 1N5819 (x3) as well as two different types of zener diode (15V [x6] and 33V [x2]) so also check Fig.4 carefully to make sure the right diode goes in the right location.

If you're using an SMD capacitor on IC1's VCAP pin, now would be a good time to solder it in place; it's labelled 47µF on the PCB silkscreen, in case a tantalum type is fitted, but a 10µF ceramic SMD capacitor is perfectly adequate.

Now fit the 28-pin DIL socket for IC1, with its notched end towards the bottom of the PCB. You can use sockets for the other four ICs but we suggest you solder these straight to the board as this will result in better long-term reliability.

Note that the four 8-pin sockets/ICs must be orientated with the notch/pin 1 dot towards the top of the board.

Fit all the TO-92 package devices next, which includes REG1 plus seven BC547 and seven BC557 transistors. Crank the leads out into a triangular pattern using small pliers, to suit the PCB pads, before soldering each one in place with the orientation shown in Fig.4.

As with the diodes, be careful not to get the similar-looking devices mixed up.

Fitting the larger components

Now you can mount regulators REG2-REG4 and Mosfets Q5-Q8, all of which are in TO-220 packages and mounted flat on the top side of the PCB. In each case, bend all three leads down through 90°, 6mm from the bottom of the package, then feed them through the PCB holes and affix the tab firmly using a 6mm M3 machine screw, shakeproof washer and nut. You can then solder and trim the three pins.

Next, fit LED1. Orientate it with the longer (anode) lead to the left and push it all the way down onto the board before soldering it in place. Now mount the

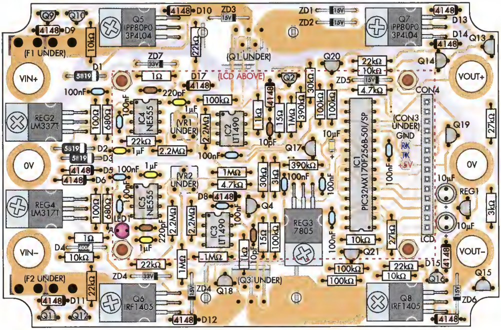
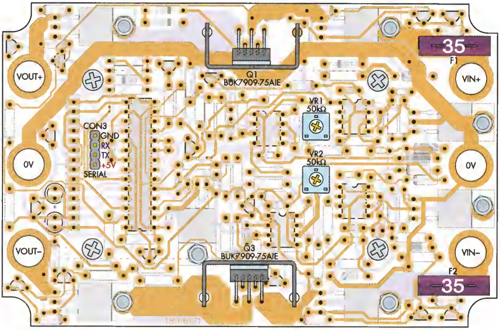


Fig.4: top side overlay diagram for the Deluxe eFuse PCB with a matching photo above. All the components are mounted on this PCB, with most on the top side. The touchscreen LCD module is mounted on top but only the dotted outline is shown, so you can see where the components go underneath. Note that there are some slight differences between this final PCB layout and the latest prototype, shown in the photos so that the high-current binding posts have more clearance.

Fig.5: an overlay diagram showing where components are mounted on the underside of the PCB. Q1 and Q3 are mounted vertically on this side so that they project down into the box and have plenty of surrounding air for cooling. The fuses are mounted on this side also, as they would foul the lid on the top side. The two trimpots also go on the bottom, so you can still access them with the touchscreen in place, allowing you to perform the calibration while watching the screen. The photo on the facing page matches this overlay.



four ceramic disc and 14 multi-layer ceramic capacitors, using the values and locations shown in Fig.4. These are not polarised.

Follow with the two 10µF electrolytic capacitors near REG1, which are polarised; the longer leads should go through the holes towards the bottom of the board. If using a 47µF tantalum instead of the SMD ceramic, it can go in now and it is also polarised, with the lead marked + on the capacitor body going in the hole towards the bottom of the PCB.

Now fit 14-pin female header CON4. To ensure it's straight, we suggest you attach the four 12mm tapped Nylon spacers that support the LCD first. These go on the top side of the board, held in by 6mm M3 machine screws fed through from the underside.

Plug the 14-pin socket into the touchscreen pin header, then feed it through the PCB and temporarily screw the LCD module to the PCB using a couple of extra machine screws.

Make sure you don't damage the touchscreen when you flip the board over and solder the header, then remove it again and put it aside until later. You can leave the tapped spacers in place

Bottom side components

The follows components are soldered on the opposite side of the board: the blade fuse holders for F1 and F2, trimpots VR1 and VR2, Mosfets Q1 and Q3 and their heatsinks and serial communication header CON3 (see Fig.5).

Solder VR1 and VR2 in place first, in the usual manner, followed by CON3.

Note that F1 and F2 may be supplied as two separate clips or one pair of clips held together with a plastic base. The type with the plastic base is easier to fit but make sure they are rotated correctly so that they line up with the silkscreen outline. Regardless, push the clips fully through the PCB and then solder on the opposite side.

You will need a very hot iron and be careful that the clips are not resting on anything which might melt while doing so. It may take some time for the solder to form proper joints so keep feeding more solder/flux in slowly until you get good-looking fillets.

Before soldering Q1 and Q3, you need to bend their leads to fit the staggered pads on the PCB.

This involves bending all five leads out slightly to the front (labelled side) of the package, by a couple of millimetres, then bending the two outer leads, plus the centre lead, forward by another 4mm. Verify that the leads fit through the holes, then loosely attach both to the inside of the "U" heatsink (as shown in Fig.5) using an M3 machine screw, shake-proof washer and nut.

You can now push the whole assembly down onto the board, with the heatsink posts going through their mounting holes and the five Mosfet leads as before. Make sure the heatsink is pushed all the way down and the Mosfet is straight, then do up the machine screw/nut tightly.

If your heatsink has solderable posts, solder these in place now; as with the fuses, this will take a lot of heat and probably some time; you may have to wait for the soldering iron to get the whole heatsink pretty warm before the solder will take.

We prefer the solderable type of heatsink but the types available from Jaycar/Altronics have anodised aluminium posts. In this case, they will just rest in the holes and the Mosfet lead solder joints will support the weight.

Regardless of the type of heatsink used, now you can solder and trim the five Mosfet leads. Make sure the solder joints are nice and solid since two of them carry the full load current. These may take a little more soldering before they form good fillets due to the large copper area connected to those pads.

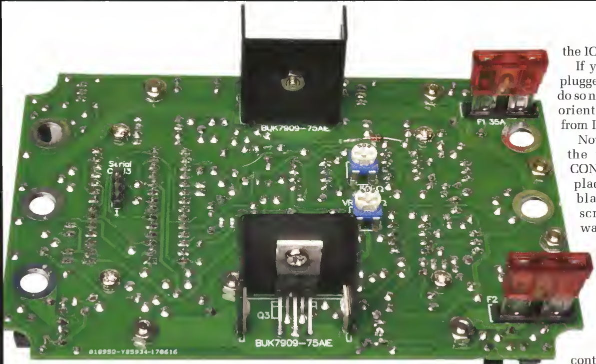
Plug in blade fuses F1 and F2 now.

Initial testing

Before going any further, it's a good idea to verify that the power supplies are working properly.

You can do this with a 12V DC plugpack, bench supply or battery. If your test power supply is not current limited (eg, a battery), use a series 5W resistor of around 100Ω to protect the rest of the circuit in case there is a fault.

First, connect the power supply ground to the 0V connection at either end of the PCB (eg, using an alligator clip against the side of the board) and the +12V output to the +IN terminal (this can also be done with a clip lead).



If the 8-pin ICs have been soldered to the board, expect around 30mA to flow, or around 20mA if they are not in circuit yet.

If using a series resistor, you can verify this by measuring the voltage across the resistor, ie, with 100Ω the voltage drop should be 100Ω x 0.02A = ~2V or 100Ω x 0.3A = ~3V. Once you've verified the current is OK, short out the resistor so the circuit can operate at the correct voltage.

Check the voltage between the 0V and +IN terminals and ensure it is at least 12V. Now measure the voltage between 0V and the anode of D1. It should be only a tiny bit less. If it's significantly lower, that suggests something is wrong with Q5 or its control circuitry.

Then check the voltage between pins 1 and 8 of IC4 (or its socket, if it hasn't been plugged in yet). You should get a reading close to 10V (9.36-10.14V). Next, measure the voltage between pin 8 of IC3 (or its socket) and 0V. You should get a reading of 4.75-5.25V. You should also get a reading close to 3.3V between pin 1 of IC1's socket and 0V.

If any of these are wrong, switch off and check for faults. If IC4 has not been plugged in yet, switch the power off and plug it in, making sure it is orientated correctly (ie, with the pin 1 dot at upper left) and then switch the power back on. Check the voltage between pin 8 of IC2 (or its socket) and the +IN terminal. You should get a reading of 8.5-9.5V.

Since IC1 is not in circuit yet, Q1 should be off and as a result, you should find the voltage at Q7's tab/mounting screw is near 0V.

To check the operation of the negative power supply circuitry, disconnect your 12V power supply and this time connect its positive output to the 0V terminal and its negative output to the -IN terminal. You should measure a similar current compared to the positive power supply.

Having shorted out the protection resistor after checking the voltage (if you're using one), check the voltage at -IN and make sure it's at least -12V, then check the voltage at the cathode of D4 which should be just a tiny bit closer to 0V.

As with IC4, you should get close to 10V between pins 1 and 8. Assuming you get the correct measurement, if it isn't in its socket yet, switch off and plug it in (again, being careful with the orientation), then switch back on. LED1 should light up at a relatively dim level. You can then check that pin 4 of IC3 is around 6.5V below V_{IN}-.

Also, check that the tab/mounting screw of Q8 is near 0V (it may be floating around). This indicates that Q3 is not conducting, which should be the case at this point.

Testing more of the circuit

To do any further testing, you will need to switch off and plug IC1 into its socket. Make sure that its pin 1 dot is aligned with the socket notch and that none of the leads get folded under the IC body.

If you haven't already plugged in IC2 and IC3, do so now, noting that their orientation is different from IC1.

Now you can also plug the touchscreen into CON4 and hold it in place using the four black M3 machine screws with Nylon washers under each screw. These will be important later when fitting the lid.

Programming the chip

If your microcontroller (IC1) hasn't already been programmed with the Deluxe eFuse firmware, you will need to program it now.

If you have a blank PIC32 chip that hasn't even been flashed with the Micromite software yet, you will have to do that before plugging it into the board since there is no provision for programming a blank chip on-board.

We expect most constructors will either have a pre-programmed chip or a Micromite.

If you have a bare Micromite chip, plug it in and hook up a USB/serial adaptor to CON3 using three or four jumper leads. CON3 has the same pinout as on the LCD BackPack. It should be labelled alongside CON3 on the PCB silkscreen.

Connect GND on the USB/serial adaptor to the GND pin (pin 4) of CON3, TX on the USB/serial adaptor to the RX pin (pin 3) of CON3, RX on the USB/serial adaptor to the TX pin (pin 2) of CON3 and optionally, the 5V supply pin of the USB/serial adaptor to the 5V pin (pin 1) of CON3.

If you decide to hook up the 5V supply lead, you can communicate with IC1 without needing to apply external power to the Deluxe eFuse board. Otherwise, you will need to provide at least 9V between the V_{IN}+ and 0V terminals.

You can then follow the instructions in the accompanying panel to set up the Micromite and load the BASIC program into it.

When you switch the unit back on, it will automatically check the V+H and

Parts list – Deluxe eFuse

1 double-sided PCB, coded 18106171, 132 x 85mm
1 ILI9341-based 2.8-inch LCD touchscreen with 320x240 pixels and 14-pin serial interface (SILICON CHIP online shop Cat SC3410)
1 UB1 Jiffy box (157 x 95 x 53mm)
1 laser-cut black acrylic lid to suit UB1 Jiffy box (SILICON CHIP online shop Cat SC4316)
2 50kΩ mini horizontal trimpots (VR1,VR2)
4 red 50A heavy duty binding posts (CON1a,CON1c,CON2a,CON2c) (Altronics P9225)
2 black 50A heavy duty binding posts (CON1b,CON2b) (Altronics P9226)
1 4-pin male header, 2.54mm pitch (CON3)
1 14-pin female header, 2.54mm pitch (CON4)
2 30A+ ATO/ATC blade fuse holders (F1,F2)
2 35A or 40A ATO/ATC blade fuses (F1,F2)
2 6021-type PCB-mounting flag heatsinks (for Q1,Q3) (element14 1317054, Jaycar HH8504, Altronics H0637)
6 M8 shake-proof washers
6 M8 spring/split washers
12 M8 flat washers
4 M3 x 12mm tapped Nylon spacers
13 M3 x 6mm machine screws
4 M3 x 8mm black machine screws
9 3mm ID shake-proof washers
4 3mm ID 6mm OD 1mm thick Nylon washers
9 M3 hex nuts
1 28-pin narrow DIL socket (for IC1)
2 8-pin DIL sockets (optional, for IC2 & IC3)

Semiconductors
1 PIC32MX170F256B-I/SP or PIC32MX170F256B-50I/SP microcontroller programmed with the Micromite Mk.2 firmware V5.0.3 or later (IC1)
2 LT1490ACN8 dual "Over-The-Top" rail-to-rail op amps (IC2,IC3) (SILICON CHIP online shop Cat SC4319)
2 NE555/LM555 timers, or equivalent (IC4,IC5)
1 MCP1700-3302E/TO 3.3V low-dropout linear regulator (REG1)
1 LM337T adjustable 1A negative linear regulator (REG2)
1 7805 1A 5V linear regulator (REG3)
1 LM317T adjustable 1A positive linear regulator (REG4)
2 BUK7909-75AIE 75V 120A 5-pin SenseFETs (Q1,Q3) (SILICON CHIP online shop Cat SC4317)
7 BC557 PNP transistors (Q2,Q4,Q9,Q10,Q13,Q14,Q21)
2 IPP80P03P4L04 30V 80A P-channel Mosfets (Q5,Q7) (SILICON CHIP online shop Cat SC4318)
2 IRF1405 55V 169A N-channel Mosfet (Q6,Q8)
7 BC547 NPN transistors (Q11,Q12,Q15-Q18,Q20)
1 BC327 500mA PNP transistor (Q19)
1 3mm red high-brightness LED, 50mA rating (LED1) (eg, Jaycar ZD0104)
6 15V zener diodes (ZD1-ZD6)
2 33V zener diodes (ZD7-ZD8)
3 1N5819 schottky diodes (D1-D3)
1 1N4004 1A diode (D4)
13 1N4148 signal diodes (D5-D17)

Capacitors
1 10µF 6.3V X7R SMD ceramic, 3216 package (1206 imperial) **OR**
1 47µF 10V tag tantalum
2 10µF 50V electrolytic
4 1µF multi-layer ceramic
10 100nF multi-layer ceramic
2 220pF ceramic
2 10pF ceramic

Resistors (all 0.25W, 1% metal film unless otherwise stated)

4 2.2MΩ	4 1MΩ	2 390kΩ	6 100kΩ	2 30kΩ	2 27kΩ	6 22kΩ	5 10kΩ
2 4.7kΩ	2 3kΩ	2 1kΩ	2 680Ω	2 100Ω	2 15Ω	2 1Ω 0.5W 5%	

Uploading the BASIC code to the BackPack

Having established a serial console connection to the PIC32 (programmed with the MMBasic 5.2 firmware) using a USB-serial adaptor, you will need to set up the display and touch panel as detailed in the February 2016 article on the LCD BackPack.

Note that the BackPack (and, if attached, the main board) can be powered from the PC during the programming process.

Once you have the touchscreen set up and working, you need to load "Deluxe_eFuse_v1.BAS" into the Micromite. Having downloaded this from the SILICON CHIP website, grab a copy of Jim Hiley's Windows/Linux "MMEdit" program.

It is freeware and available from www.c-com.com.au/MMEdit.htm For Windows, download the setup file called MMEdit.exe and run it. It will work on any Windows version since XP.

Run MMEdit and open the BASIC file mentioned above. Next, ensure the "Auto crunch on load" option in the Advanced menu is selected and set up the COM port to communicate with the Micromite by selecting the "New..." option under the Connect menu. You can then click the "Load and run current code" button, right-most in the toolbar under the menu (with the icon that looks like a blue stick figure running while holding a torch).

You should get a progress dialog and the upload will take around 30 seconds. If it fails, close this window and re-check the COM port settings; make sure you don't have the port open in another program.

Once the upload is complete, the MMChat console window should automatically appear. You can then type in "OPTION AUTORUN ON", press enter, then execute the "RUN" command to start the program. The unit should then start operating. Assuming it does, unplug the USB lead and proceed with the remainder of construction/set-up.

V-L voltages and verify that they are in the expected ranges. We've already verified they are in the required ranges so it should boot up normally but if a problem is detected, you will get a message on the screen indicating the problem and you can then switch off and check the circuit for faults.

Calibration

The Common Mode Rejection Ratio (CMRR) of the two differential amplifiers must be optimised to give correct current readings and trip levels. This is relatively easy but requires some test loads.

Power resistors are suitable; for example, a 33Ω 5W resistor can be used to calibrate CMRR in both channels, using two 12V plugpacks or batteries (or a ±12V bench supply).

Power up the unit by applying +12V to V_{IN}+ and connect the 33Ω resistor between V_{OUT}+ and 0V, with a multimeter connected in series and set in DC current measurement mode. Try to let the resistor hang in free air since it will get quite hot during this procedure. Set VR1 to its midpoint and set the trip current to maximum, then switch on the output. Adjust VR1 until the reading on the LCD screen is close to the reading on your multimeter.

Then disconnect the load and check that the current reading falls to 0A. If not, rotate VR1 as little as possible to get a reading of 0A.

Re-apply the load current and check that the reading is still correct. If not, use the software current scale calibration (see below) to correct it. If you have another different value power resistor, you can connect this and verify that the current reading is still correct.

The procedure to calibrate the negative channel is similar except that you will need to apply +12V to V_{IN}+ (for the digital circuitry to operate) and -12V to V_{IN}-.

You can provide the -12V supply using a second 12V plugpack (with floating [unearthed] output) or battery, as long as you connect its positive terminal to 0V and negative terminal to V_{IN}-.

Software calibration

While it isn't strictly necessary, you can also calibrate the voltage measurements, to compensate for variations in resistor values, regulator outputs and so on.

To do this, apply power as above but connect your multimeter between V_{IN}+ and 0V, in DC voltage measurement mode. Access the calibration menu by holding your finger in the centre of the touchscreen for several seconds.

You can then use the + and – buttons to adjust the V_{IN}+ reading to match what you're getting on your DMM. Then connect the DMM between V_{IN}- and 0V and adjust the V_{IN}- reading in the same manner.

You can also use this screen to zero the current readings for both channels or adjust the current scaling factor in software using the adjacent +/- buttons. This should only be necessary if you can't use the CMRR adjustment to get accurate current readings at different current levels.

In this screen, you can also change the default screen brightness, whether the screen backlight dims and eventually turns off automatically and if so, the duration of touch inactivity required to activate the automatic dimming.

This can be useful to reduce the extra current drawn from the supply, and resulting extra dissipation in the case when using the unit for extended periods. All these setting are stored in flash so you only need to set them once.

Finishing the assembly

The lid requires a large, straight, rectangular cut-out for the LCD touchscreen to fit through, four mounting holes for the LCD module plus six large, profiled holes for the high-current binding posts.

Since cutting all these accurately would be time-consuming and difficult, we can supply a laser-cut replacement lid made from black 3mm acrylic that already has all these holes cut out precisely.

The plastic the lid is made from is matte on one side and glossy on the other. Since it's symmetrical, you can use it either side up, so you can choose how you want the front panel of the unit to look.

The rest of the assembly instructions will assume you're using the pre-cut lid.

Next month we'll go over the remainder of the instructions, give some sample screen grabs from the software and describe its operation. SC