Digital Slave Unit: Assembly Notes: *Draft document*

0 Contents

1 Features of Circuit

This slave unit has all the same features as my previous design, and the *Firefly 2*. In addition, it has the 'digital camera facility' I have described previously. The circuit board fits in a small plastic case, identical to the *Firefly 2* case. One additional feature is the presence of a link, on the circuit board, which can be used as an on/off switch. In normal use, the slave unit consumes so little current that it does not need an on/off switch. However, if you are intending not to use it for a period of six months or longer, then you can extend the battery life by removing the link. The link can be seen in the photograph that accompanies step 7 of the assembly instructions below.

2 Notes For Kit Builders

2.1 Other documentation

The document you are reading now is an advance draft copy. It is intended only to describe how to *build* the units, not how to operate them. You will also need to read…

A High-Performance Flashgun Slave Unit, online at
caves.org.uk/flash/slave2005-1.pdf

Slave Unit for Digital Cameras, online at
caves.org.uk/flash/j57.digislave.ebook.pdf

2.2 Contents of Kit

The kit of parts is shipped with a copy of the circuit diagram. *There are no other instructions.*

2.3 Suitability for Purpose

If, after reading these notes, and inspecting the kit, you decide that it will not be suitable for your purposes, you may return it unused within 30 days (and at your own expense) for a refund.

Professionally-built versions of this product are available from ***Firefly Electronics***, 61 Princes Street, CARDIFF, CF24 3SL. +44 (0)29 2049 8112. **Info@fireflyelectronics.co.uk**, **www.fireflyelectronics.co.uk**. The Firefly product is *based* on this design, but it is not an exact copy.

2.4 Warnings

Before ordering or building this kit of parts you are advised to read this note and understand it!

This kit of parts is intended for cave photographers who, as a matter of habit, work with flashguns that they have re-packaged for use underground (perhaps with different housings and different connectors). It is not primarily intended for use with commercially available flashguns, although you can fit a hot-shoe adapter to the slave unit if you wish to use it with such guns. However, you should be aware that hot-shoe adapters are not intended for use with 'dedicated' flashguns; that is, flashguns that are designed for use with a particular camera and which offer special features when used with such a camera.

I have designed this slave unit as an aid to cave photographers. I have published the circuit diagram for your personal use. Although I have not published the software, I am making available programmed and tested micro-controller chips. Those items are sufficient for you to built yourself a slave unit.

I am also making available printed circuit boards, to save you having to design your own; and I am providing a kit of parts to save you having to go through the tedious process of ordering components. Making these additional items available to you should not be taken as any sort of guarantee that the slave unit you build will work, or will be suitable for your purpose. Please note, in particular, that the instructions I supply are not intended to be comprehensive. They are not intended as a substitute for your professional electronics knowledge and your high degree of skill in the assembly of micro-electronic equipment using surface-mount devices. The basic material that I am intending you work from is the circuit diagram, component list and PCB layout – and those alone. As a further caution, it must be said that if you are not already highly-experienced in the hand-soldering of surface-mount and other miniature components then you would be extremely foolish to practice on this unit, because you will undoubtedly ruin it and thereby waste your money.

I *know* the circuit works; I *know* the software works. If yours does not then it is almost certainly your fault – you have damaged something building it, or you have not built it correctly, or it is not suitable for your purpose. And that is for *you* to find out and *you* to deal with. I am not intending to put myself under any obligation to you – I am publishing an electronic design and you are choosing to use your specialist electronic design and construction skills to turn it into a product for your personal use. That is *your* choice!

Of course, consumer legislation means that if I sell you something, I cannot avoid some degree of responsibility, so if you think a component is damaged when you receive it then I will replace *the component* free of charge. If you decide that the kit is not suitable for your needs then you can return it *unused* within 30 days, for a refund. If you want to dispose of the product after use, you can return it to me (at your own expense) for that purpose.

2.5 Surface Mount Devices

It is not really my intention to offer guidance on how to deal with SMDs, since I am not in a position to recommend a course of action to you: *you* are the expert in hand-soldering SMDs, not me! However, you might find that it helps if you glue the small Cs and Rs before soldering. (not the large tantalums, or the switches, or LED). Do not use superglue, or you will end up gluing your finger to the board.

3 Bill of Materials

The BOM is shown on the next page. The last few shaded columns are intended as a guide to people who wish to source their own components, and need to know if they will fit the board. From the information given, you can visit the web sites of the distributors and – in most cases – download the manufacturer's datasheet.

Please note that the parts supplied in the kit may not be from the manufacturer listed in the BOM, as alternatives may have been sought out for reasons of cost or availability. It is not my intention to keep the shaded columns up to date.

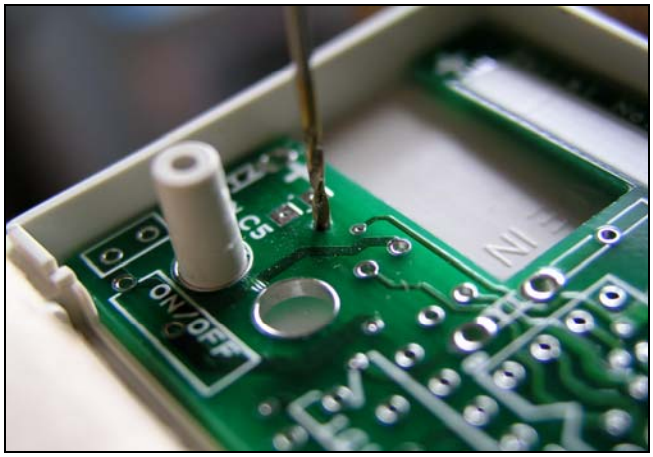
3.1 Legend for BOM

♦	Part not required for 'basic' slave unit
SMD	Surface-mount device
TH	Through-hole device
TS	Top-soldered device. The device is soldered on its 'component' side, not on its 'pins' side
✶	See notes: 1. An alternative part may be provided. See assembly notes. 2. Spacer beads are required for use with C4 3. An alternative part, BPW41N, may be provided. See assembly notes.

Qty	Kit Check	PCB Ident	Board side	PCB mounting	Packing location	Schematic Part Name (from circuit diagram)	Part Description	Supplier's Stock Code	Supplier (if not Farnell)	Manufacturer	Manufacturer's Part Number
1	♦	C5	Top	SMD	Bag 4	MLC 1U0 SMD	Multi-layer Ceramic Capacitor	422-7190		Multicomp	T0805F105ZNT
2		R1, R9	Top/bot	SMD	Bag 3	10M SMD	Metal Film Resistor, 1%	308-8261		Vishay	2312 155 11006
2		C1, C2	Bottom	SMD	Bag 3	TN 22U 6V3 SMD	Tantalum Capacitor, Low Leakage	967-026		Multicomp	MCCTC226M006
1		C3	Bottom	SMD	Bag 3	MLC 10N SMD	Multi-layer Ceramic Capacitor	422-7153		Multicomp	U0805R103KNT
2	♦	SW1, SW2	Bottom	SMD	Bag 3	KEY 4MM SMD	Key Switch 7914J-001-000 Bourns	535-930		Bourns	7914J-001-000
1	✶1	C4	Top	TH, 0.6″	Bag 3	MPETP 400V 100N *to amend*	Polyester Film Capacitor, 400V d.c.	548-182		EPCOS	B32562J6104K
1		LK1	Top	TH	Pot	PINS 3	Pin Header	512-047		Harwin	M20-9990305
1		LK2	Top	TH	Pot	PINS 2	Pin Header	512-035		Harwin	M20-9990205
1		SK1	Top	TH	Pot	TERM2 *to amend*	Screw Terminals	**To be added**			
2		Y1, Y2	Top	TH	Pot	TAB 6MM RT	Faston Tab 6.3 X 0.8mm	362-5590		JST	TAB3867B68(MSTIN)
2		R10, R12	Bottom	TS, 0.2″	Pot	3K3	Metal Film Resistor, 1%	332-112		Vishay	MRS16T 3K3 1%
1	♦	R11	Bottom	TS, 0.2″	Pot	470R	Metal Film Resistor, 1%	332-010		Vishay	MRS16T 470R 1%
1	♦	R13	Bottom	TS, 0.2″	Pot	820R	Metal Film Resistor, 1%	332-045		Vishay	MRS16T 820R 1%
2		R2, R3	Bottom	TS, 0.2″	Pot	100R	Metal Film Resistor, 1%	331-934		Vishay	MRS16T 100R 1%
1		R4	Bottom	TS, 0.2″	Pot	1M0	Metal Film Resistor, 1%	332-410		Vishay	MRS16T 1M 1%
1	♦	R5	Bottom	TS, 0.2″	Pot	10R	Metal Film Resistor, 1%	331-818		Vishay	MRS16T 10R 1%
1	♦	R6	Bottom	TS, 0.2″	Pot	33K	Metal Film Resistor, 1%	332-239		Vishay	MRS16T 33K 1%
2		R7, R8	Bottom	TS, 0.2″	Pot	100K	Metal Film Resistor, 1%	332-290		Vishay	MRS16T 100K 1%
1		FS1	Top	TH, 0.2″	Pot	FUSE MF 0.1A	Multifuse, 0.1A 60V	772-252		Bourns	MF-R010
2		---	Top	Press-fit	Pot	---	Self-locking spacer 6.3mm, 4mm fix hole	861-583		Harwin	R6006F00
1		---	Top	Press-fit	Pot	---	AAAA battery contact	302-9803		Keystone	204
1		---	Top	Press-fit	Pot	---	Jumper 10mm blue	321-8491		Harwin	M7687-05
1		---	Top	Press-fit	Pot	---	Jumper red	**To be added**			
2	✶2	---	Top	Fit to C4	Pot	---	Plastic spacer bead			*Local shop*	
1	✶3	D1	Top	TH	Card box	SFH206K	Photodiode SFH206K or equiv.	212-714 or 327-438		Infineon / Vishay	SFH206K or BPW41N
1	♦	D2	Bottom	SMD	Card box	LED RED/GRN SMD	KA-2734SRSGC	72-8680	Rapid	Kingbright	KA-2734SRSGC
1		IC1	Top	TH	Card box	TLC27L2CP	Op-Amp	401-717		Texas Instr.	TLC27L2CP
1		---	Top	TH	Card box	---	8 pin IC socket	424-2348		Multicomp	2227MC-08-03-18
1	♦	IC2	Top	TH, socket	Card box	PIC12F675	Programmed part from David Gibson	413-6883	Gibson	Microchip	PIC12F675-I/P
1		OPT1	Top	TH	Card box	TLP3022	Optocoupler Triac Driver	620-750		Toshiba	TLP3022-S
1		Q1	Top	TH	Card box	Z00607MA	Triac	323-9433		ST Micro	Z00607MA
1					Envelope		Printed Circuit Board	SLV051	Gibson		
1					Bag 1		Solder	629-431		Multicore	692431
1					Bag 1		Cable Tie, 100mm	151-271		Hellermann Tyton	T18R-RED(100 PACK)
1					Bag 1		Grommet,9.5mm (6.4mm hole)	152-332		Unbranded	PV5 GROMMET PK 100
2					Bag 1		Velcro Coin Loop	957-306			VC11L-804-R5(22.0)
2					Bag 1		Velcro Coin Hook	957-290			VC11H-804-R5(22.0)
2					Pot		Punched Roundels, 6mm x 0.7mm			*Local shop*	
1					Bag 2		Yellow Polyester Labels	44A-LA7778	CILS	CILS	44A-LA7778
3					Outer		Battery, Button Cell 1.5V "357" 180mAh	300-860		Varta	V357
1					Outer		Clipper Box, 60x40x18.5 White, & Acrylic Infrared Panel	001111 & 001150	West Hyde		
½					Outer		Phono Lead & Plug	371-2205		Combined Precision Comp's	AV00243
1					Envelope		A6 Laminating Pouch	VLP17	Viking		
1					Outer		Circuit Diagram and Instruction Sheet		Gibson		

4 Assembly Instructions

1. Place PCB on mounting pillars in the case (make sure the orientation is correct – serial number panel should be visible). Using a bench drill (to ensure correct alignment) and a 1mm drill bit (no larger!) drill out the three holes marked on the PCB overlay (see step 14), drilling on through the case. It is important that you do not drill the wrong holes, nor use too large a drill. (On early versions of the board, the silkscreen indication of which holes to drill is on the wrong side of the board. Do not mount the board upside down to drill the holes or they will be in the wrong orientation).



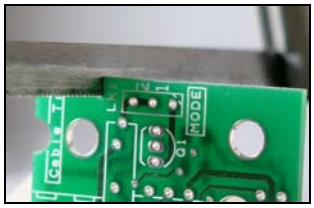
2. Remove PCB and drill the outer two pilot holes you have made in the case to 6.5mm; dril lthe other one to 4mm. You can see these holes in the photo in step 6). Make sure they are clean, with no burrs.

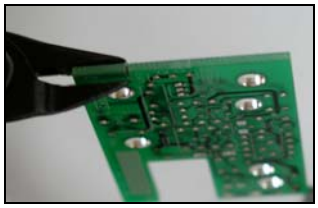
   Snip and mount the main yellow sticky label (see step 17 on page 2 of these instructions for ***important*** information)

   Crop the two studs from the top of the case, as shown in the photo below. With some versions of the PCB these studs may foul against components mounted on the circuit board.

3. There are two mistakes on the early PCBs, which need some re-working, which you should now perform. The PCBs that are affected are labelled **Version A-07, PCB SLV051-6**.

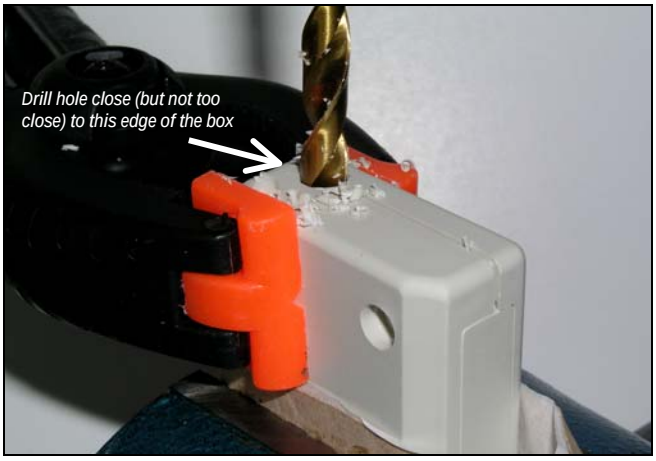
   You need to cut a notch in the edge of the PCB. With the board positioned as shown below left, cut a 3mm slot with a junior hacksaw. Then, using a sharp and strong pair of cutters snip the board (below right) to remove a thin strip, as shown in the photo in step 4. This is to avoid the edge of the board fouling on the grommet (see photo in step 6).



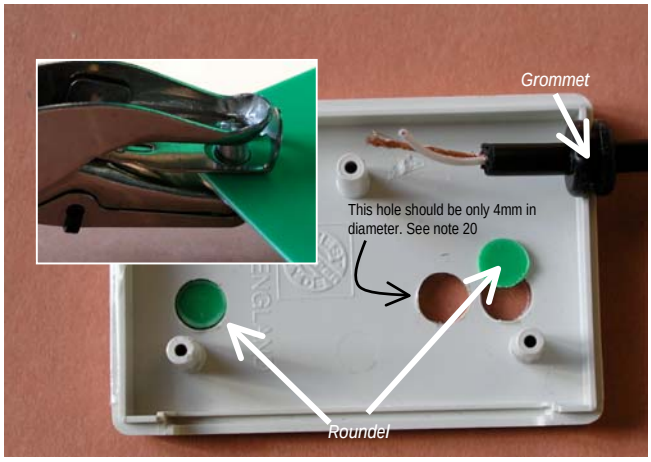


4. The terminal block SK1 supplied with the kit has its pins on a 3.5mm pitch, whereas the holes on the PCB are at 5mm. You need to drill a 1.5mm hole in the PCB to accept one of the pins of the terminal block, as shown below. You must ensure that you do not foul the track on the reverse of the PCB, or you will be in danger of shorting SK1 when you solder it.

5. Drill a 6.5mm hole for grommet, so that it straddles both halves of the case. Make sure that you drill the hole in the correct side of the box. You will find it easier to drill the hole if you clamp the sides of the case together firmly with a tool clamp. Drill the hole as close to the edge of the board as is consistent with being able to fit the grommet properly, otherwise the grommet will foul against the PCB.



6. If the two 6mm 'roundels' are not included in the kit, you will need to punch these, using a single-hole paper punch, from a sheet of 0.7mm (or slightly thinner) plastic. You can use cardboard, but this runs the risk of getting damp and disintegrating. Fit as shown below. It is ***important*** that the holes and the plastic discs are all burr-free, or the key-switches on the PCB will be liable to jam, and prevent the unit from operating properly.

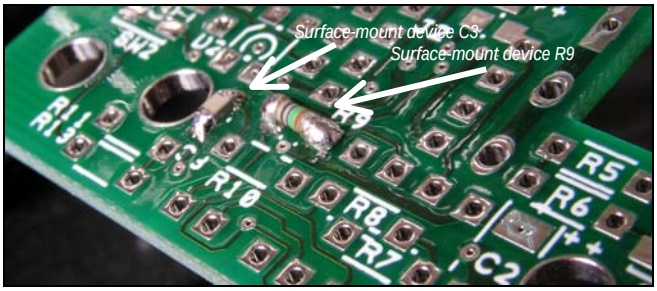


7. Fit SMDs C5 *[part not fitted for 'basic' slave unit]* and R1 on the top side of the board. Fit the remaining SMDs on the bottom side of the board. If you have never soldered surface-mount devices before, this board is not the place to practice since you will inevitably damage the board or the components.

   Some of the SMDs will be easier to solder than others. The very small SMD capacitors are particularly easy to damage since they come into direct contact with melted solder during soldering. The component is exposed to potentially damaging mechanical stress caused by the sudden temperature change, and can also be subject to silver migration and to contamination by the flux. In the worst case, the large temperature gradient can cause mechanical damage such as cracking of breaking of the devices. Standard advice includes the following tips: i) Control the temperature of the tip of the iron (300C). ii) use a small (<20W) iron. Iii) use a large tip (3mm). iv) melt the required amount of solder onto the tip of the iron first. v) pre-heat the part so that it is at least 150C. vi) after soldering, cool the component gradually.

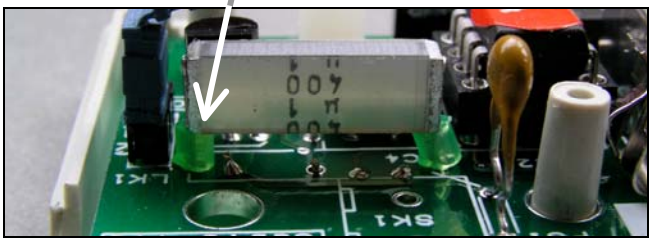
8. Fit the resistors on the bottom side of the board. They should be "top-soldered" – that is soldered on their component side, not on their pins side. Note that the component side is the *bottom* side of the PCB – the side with the **large square pads** – so 'top' soldering means soldering on the 'bommom' side of the board! Crop the leads flush on the top side of the board. These resistors are on a 5mm (0.2″) pitch, which is the minimum recommended by the manufacturer. Do not bend them to a smaller pitch than this, or you may damage the case of the component.

   As an aid to inspection, make sure that all the resistors are orientated the same way round – i.e. the colour codes all read top to bottom or left to right. It is also a good idea to use a meter to check the value of the resistors as you fit them, because the bars are not always easy to read.



9. Fit the remaining components. You will find it easier to fit the smaller ones first. Do not fit IC2 directly to the board – use the socket provided.

   Leave C4 until last. C4 is either a dipped-epoxy part, orange in colour, or an open-frame part, grey/silver in colour. If the former, the component should be mounted bent, so that it fits in the case. If the latter, it should be mounted using the two green plastic beads supplied (see photo below), so that its base is clear of the PCB. This is **important**, and is to prevent the capacitor (which is un-insulated) shorting onto the circuit board.



10. An alternative part for D1 may be supplied:

    SFH203K (clear plastic) This is sensitive to visible and IR light. Use this if you need additional sensitivity, or if you wish to use the unit underwater.

    BPW41N (black plastic) Sensitive to IR light only.

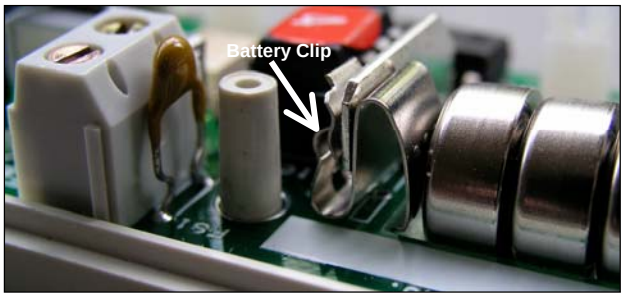
    Either part can be used with the black infrared-transparent window but, if you need to utilise the additional sensitivity of the **SFH203K** to visible light then you must use a transparent window, which you must construct.

    If you are building the unit without the digital camera facilities then you should place a blob of solder across the link J2 on the bottom side of the board. If you are adding the digital facilities to an existing unit, you need to remove this link with a de-soldering tool.

11. Scrub both sides of the board in warm water to remove any flux. This is ***very important***, as the flux in the water-based solder provided in the kit will harden to an immovable and *conductive* residue if it is left to harden. Scrub the board ***gently*** or you will damage the key switches, which are delicate.

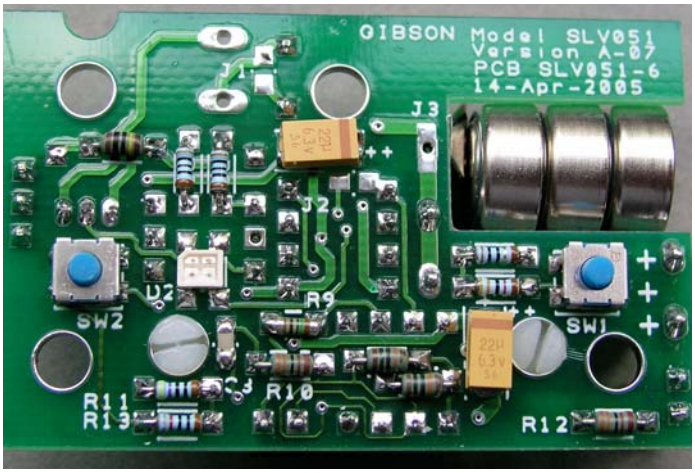
    If you do not complete assembly of the board in one stage, you should scrub the board when you finish each soldering session.

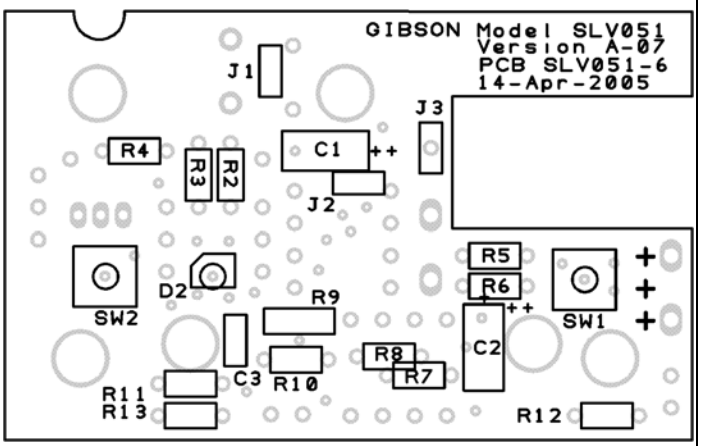
    Inspect the board carefully. Make sure all the components are in the right places; that the polarised components are orientated correctly (D1–2, C1–2, IC1–2, OPT1, Q1) and that there are no solder bridges or flux residues.



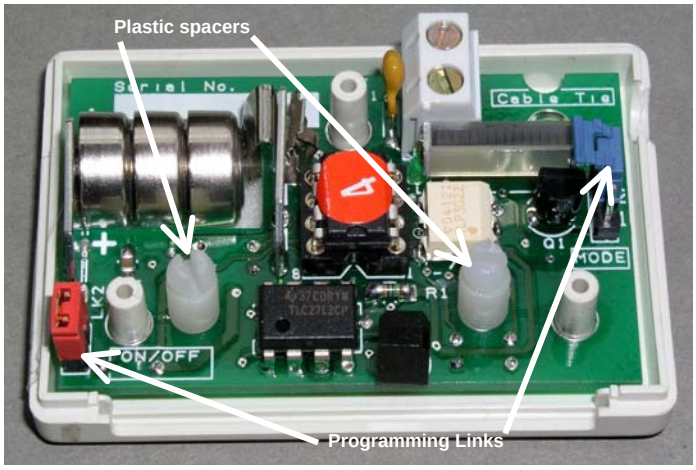
12. Fit the plastic spacers, battery clip (to terminal Y1 – see photo above) and programming links. On some issues of the PCB, the battery clip cannot be positioned centrally as it fouls against the socket for IC2.

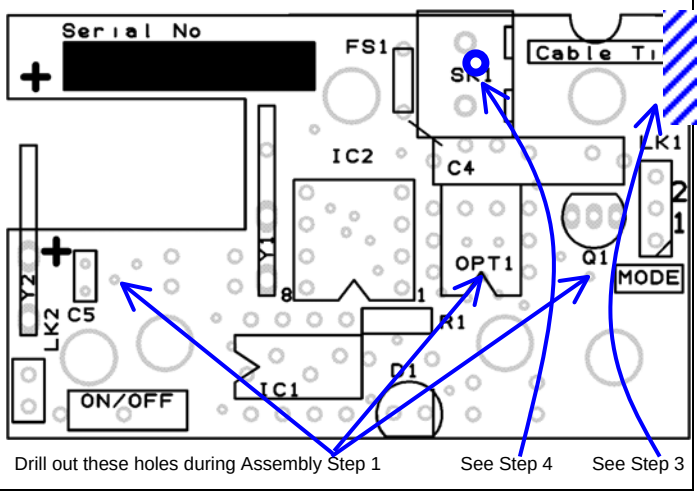
13. Photo and component overlay of bottom side of PCB





14. Photo and component overlay of top side of PCB





15. Thread the grommet onto the phono lead. Bare the ends of the phono lead, tin them, and attach the lead to the terminal block. Secure it with the cable tie.

16. Fit the batteries and the programmed micro-controller chip.

    The slot for the batteries on the early PCBs, labelled **Version A-07, PCB SLV051-6**, is too large, and if you push the batteries all the way in, they will sit too low and force the PCB upwards. This may prevent one of the pushbuttons from working properly, so you should not push the batteries in too far. You may like to glue a small piece of foam or cork to the case, to prevent this occurrence.

17. Fit the remaining yellow sticky labels. There are four labels in the set supplied (see diagram below and pictures, right), which are printed on a sheet of waterproof sticky-backed polyester

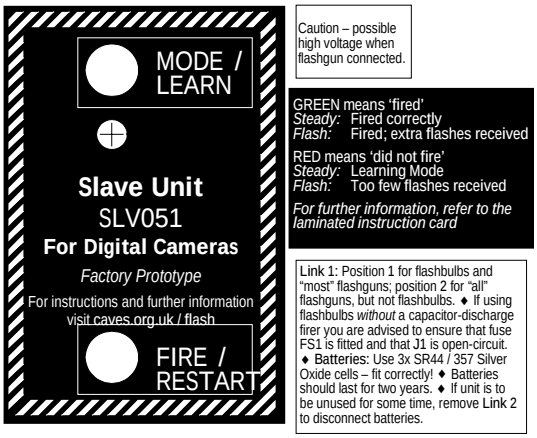
    The large label should be fitted at step 2, earlier, to the outside bottom of the case. Before fitting it, you should punch a hole (e.g. using a single-hole paper punch) where the LEDs need to shine through). This is marked with a '+'; on the label. To assist your punch in making a clean hole, place a piece of thin card behind the label and punch through both items at the same time.

    This hole requires a small piece of thin transparent plastic to seal it. You could cut a small square from a food container, or use a doubled-over piece of sticky tape.

    The smaller mainly-black label goes on the outside top of the case.

    The 'reminder' label about links and batteries goes on the inside of the lid.

    The 'caution' label should be affixed to C4.



The set of labels supplied with the kit is printed on a sheet of waterproof sticky-backed polyester

18. Cut the 'User's Instruction sheet' from the next page of this document and place inside the A6 laminating pouch.

    If you do not have access to a laminator with which to seal this pouch, use a slow iron on top of a thin cotton handkerchief to heat the sheet of plastic. When it has sealed, carefully run the edge of the iron around the edges of the plastic to ensure the seal is complete.

    Punch a hole in the laminated sheet to attach a lanyard.

19. **Waterproofing:** clearly, the unit is not waterproof, and so for use underground you will have to consider how you can protect it from water.

    The circuit is very sensitive to moisture, and it must be kept dry. You may wish to conformally-coat the PCB. If you do so, make sure you use a compatible product that is recommended for the purpose.

    You may wish to seal the unit with tape before you use it underground.

20. ***Miscellaneous Points***

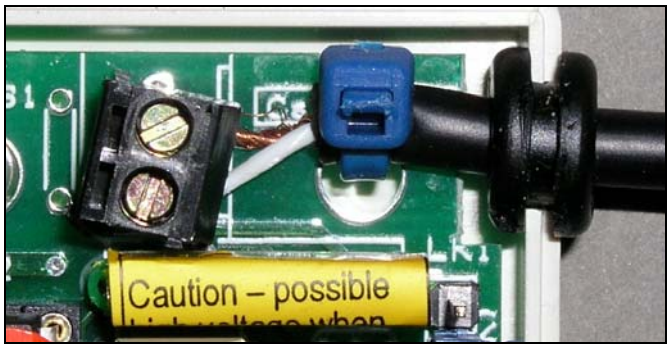
    The yellow label on the outside of the case does not stick perfectly. One of the factors that affects this is the size of the transparent window you use (as described in step 17). For this reason, it may be advisable to drill the hole for LED in the case to a smaller diameter than the other two. (As noted in step 2).

    As an alternative to using the solder provided with the kit, and needing to wash the board in water, you could use your own solder, provided it is a *low-residue* solder that does not require cleaning. Do not use 'traditional' solder that leaves a residue, because you will be unable to inspect the board properly afterwards. Using a solvent to remove flux residues is difficult in a home environment and can – if you are not experienced – merely distribute a sticky mess over the entire board.

A view of the completed unit. The infrared window is to the left

Bottom view of the completed unit. The infrared window is to the left

View showing the position of the two internal information labels.



View showing the alternative terminal block (see step 4) and the slot in the prototype PCB, which is cut to accommodate the grommet (step 3).

5 Information Sheet

Trim this information sheet, fold it in half and place it inside the A6 laminating pouch. Alternatively, if you print it twice – using both sides of the paper, you will end up with a double-sided A6 sheet, which is better suited to laminating, because a folded sheet relies solely on the glue around the margin to hold the pouch together. When printing the second side, make sure that the bottom of this sheet – marked BOTTOM – feeds through the printer first. It might take several attempts to get the paper to line up exactly right. See instruction step 18 on the previous page for further information about laminating.

BOTTOM

Specification

Run Mode Settings

Standard	fires after one pulse
Multi-Pulse	fires after a number of pulses
Delayed Fire	fires after a short delay

Learn Mode
Allows unit to be programmed to one of the above settings, or to 'learn'; from a camera or flashgun.

Flash Pulses

Minimum delay between pulses	2ms
Maximum spread of flash pulses	1s
Delayed-Fire delay	2ms *

* may be user-adjustable in future versions

Controls

Restart/Fire Button	for Manual firing
Mode/Learn Button	for programming
LEDs to show…	Operating Mode
	Firing Status
	Low Battery

Power Requirement
3 × SR44 / 357 Sliver Oxide cells (1.5V, 170mAh)
Standby current <10µA
Battery life at least 2 years / 20,000 ops.
If unit is to be unused for some time, remove Link 2 (inside case) to disconnect batteries

Output Rating

Trigger type:	electronic (triac)
Switching voltage:	400V max
Switching current	10A max (peak)
	800mA max continuous*

* or 100mA if resettable fuse (FS1) is fitted, If using flashbulbs without a capacitor-discharge firer you are advised to ensure that FS1 is fitted and that J1 is open-circuit.

Output Setting (Internal Link LK1)
Position 1 for flashbulbs and "most" flashguns; position 2 for "all" flashguns, but not flashbulbs.

Physical Information

Dimensions	*to be inserted*
Weight	*to be inserted*

David Gibson,
12 Well House Drive, LEEDS, LS8 4BX
david@caves.org.uk, **caves.org.uk / flash**

Status LEDs

After receiving flash pulses, the LEDs will light for 8s with the following meaning

GREEN means 'fired'

Steady Green	fired correctly
Flashing Green	fired, but extra flash pulses were received

RED means 'did not fire'

Steady Red	Learn Mode
Flashing Red	did not fire; too few flash pulses received

After Pressing the Restart/Fire button, the LEDs will light as follows

1) Brief simultaneous flash of Red and Green LEDs. This is part of a self-test feature. One action during this time is that the battery voltage is checked.

2) The red LED will blink steadily a number of times, to indicate the number of flash pulses the unit is programmed to detect. For the Standard setting this is one, of course.

Or The Red LED will give a series of double-flashes to indicate the number of ms delay in Delayed Fire setting.

3) Finally, **the** Green LED will light for 8s to indicate that the unit has fired.

If, during this time, the Red LED gives a very brief double-flash, repeated with 2s interval this is '**low battery**'.

At power up: When you insert a battery the red and green LEDs will blink alternately. **The unit will initialise to Standard Setting.**

Software Version

Software version slv-2v1

This is a prototype for evaluation 1) "Permanent Learning Mode" not implemented. 2) Low battery calibration needs refining – voltage threshold may not accurate. 3) Other minor issues known about.

Identification

PCB Serial No _____
Software Serial No _____
FS1 fitted? _____
Photodiode *SFH* (Vis./IR) | *BPW* (IR)
Window transparency Visible / IR

Restart / Fire Button

Allows you to:

1. **Fire the slave unit manually.** This is useful if you need to test the connection to your flashgun. The slave unit will fire when the button is *released*.
2. **Restart the operating program.** If you get muddled during programming, this button resets the microprocessor. Restarting the program like this does not change the operating mode.
3. **Check the Run settings.** When you release this button, the slave unit fires. The red and green LEDs then blink in a coded pattern to indicate the run settings. Finally, the green LED lights for 8 seconds which is the indication that the slave unit has fired (see Box: **Status LEDs**).
4. **Reset the operating mode**. If the Mode button is held down whilst the Restart/Fire button is pressed then the unit will reset the operating mode to Standard. For this operation, the slave unit is not fired when the Restart/Fire button is released.

Low Battery Calibration

You must use a fresh set of batteries for this exercise. If in doubt, check the voltage.

1. Press the Restart button to activate the Mode button.
2. Whilst the Green LED is lit, press and hold the Mode button for at least 10s.
3. Eventually, the Green LED will give a brief double-flash. Release the mode button.
4. The battery voltage has now been recorded. The low battery threshold is set to 50% of this value, so with three 1.5V cells, the threshold will be 3.0V.

You cannot erase this value – you can only record a new one. If you record with a partially flat battery you will move the threshold down to below 3V.

The low battery indication does not inhibit the operation of the slave unit – it is just a warning.

Mode Button

Allows you to:

1. **Enter Learn Mode**. The Mode button is only active when the LEDs are reporting Status information so, if they are both unlit, just press Fire/Restart so that the Green LED lights. Then press Mode to enter Learn Mode, which will cause the Red LED to light.
2. **Program the number of flash pulses.** In Learn Mode: Each press of the Mode button will be counted and the unit will use this data for future firings in Run mode. Pause for unit to revert to Run mode (Red LED will extinguish).
3. **Count the pulses from a flashgun.** In Learn mode: fire your camera or flashgun. The number of flashes will be counted and the unit will use this data for future firings in Run mode.

The number of flash pulses the unit will count is limited only by the fact that the pulses must be spaced by at least 2ms, and that they must all happen in a one second period.

After the unit has counted the pulses the Red LED will blink for 8s to indicate that "flash pulses were received, but the unit did not fire". When the LED goes out, the unit will be programmed and ready to work in Standard or Multi-Pulse mode as appropriate. You can check the setting by pressing the Restart/Fire button as described above.

4. **Set the operation to Delayed Fire.** If you do nothing at all in Learn mode it will time out after 15 seconds. The unit will be set to Delayed Fire and the LEDs will show this by blinking the status information.
4. **Set a Permanent Learning Mode,** where each time the unit is triggered, it counts the number of flash pulses received, and stores this to use the next time the unit is triggered. *THIS MODE IS NOT IMPLEMENTED ON THIS ISSUE OF THE SOFTWARE.*

Punch this hole to attach a lanyard