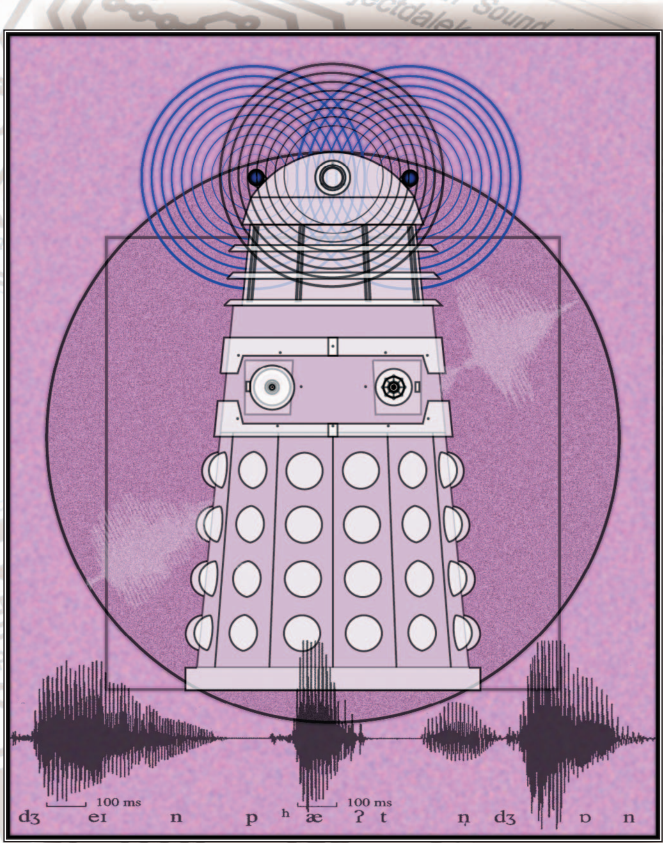
DALEK



VOICE MODULATOR ASSEMBLY MANUAL

www.projectdalek.co.uk

Version 2.07 - March 2011

(NOTE: To update printed copies from v.2.06, replace pages: 2, and 20 only).

by Fenris

with invaluable help and support from Bertus.

Layout design by John Darley.

The Project Dalek Voice Modulator is intended to be an 'open source' project for Forum members. All information is given in good faith and may be used to build a voice modulator from scratch or from one of the parts/kit options which may be offered from time to time.

All we ask in return is that you don't exploit our collective efforts by auctioning or selling built up kits at vastly inflated prices. Such profiteering is not in the spirit of this project.

www.projectdalek.co.uk *www.projectdalek.com*

Read This First!

This preface contains important information on certain options and construction of the voice mod. Please pay attention and read this in conjunction with the rest of the manual. You will then have a full picture of what is entailed and can make decisions based on the details herein. You have been warned!

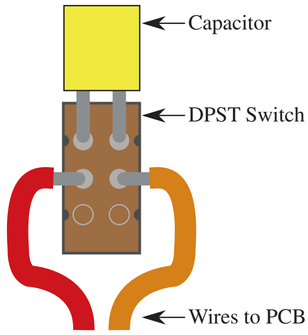
The first important thing to note is the control of the modulation rate. This has been re- designed such that there are twelve positions on the modulation rate control that relate to the twelve frequency's used throughout Dalek film and TV history. Please note that due to component tolerances the frequency you want may sit either side of the theoretical position. Two of the frequency's are only attainable by adding a 1uF capacitor to P1 on the PCB.

C11 and P1 are the timing capacitors for the modulation rate. As such, both need to be selected for the narrowest tolerance you can get. Ideally 5% should be the maximum you go for. The reason for this is that if the tolerance is higher, i.e. 20%, the actual component value could range from 0.8uF up to 1.2uF which will effect the frequencies. At worst this could mean that the voice mod doesn't sound very Dalek-like. In the components list, item 10-5836 is the preferred choice.

It is suggested that if you want to take advantage of all the range that is available then the capacitor in question is fitted as follows...

First of all you need a switch that is described as DPST. This can be a toggle or slide switch. The initials stand for Double Pole Single Throw. Basically the switch is two switches in one and is either on or off. The switch will have four terminals. These will be in pairs side by side. There will be a pair in the middle of the switch and a second pair nearer one end. Simply attach the capacitor by soldering one leg to one of the terminals of the pair near the end and then the other leg to the remaining terminal of the same pair. This will leave the middle pair of switch terminals. Now fit two wires from the C11 or P1 header on the PCB, one wire in each hole, to the remaining switch terminals, one wire to each terminal, and your sorted.

General arrangement of switched capacitor



There is an alternative set-up to the frequency control. This is based on the original unit and if so desired can be used by the builder. With this set-up only a small portion of the range of the 1M pot gives a usable frequency that can be considered sufficiently Dalek like. Components to change/remove are -

VR1 10K Lin change to 1M Lin

R14 8.2K change to 1K

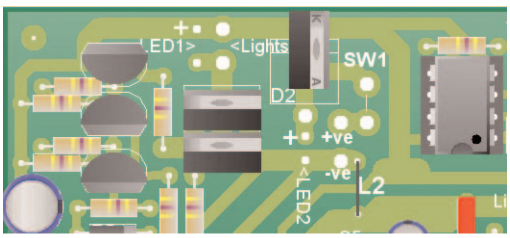
R35 Not fitted

C11 1uF change to 100nF

1. Although throughout the 'Sequence of Assembly' and a few other instances, the PCB is shown in one piece. It is intended that the tone control circuit is separated from the main PCB. You can, if you wish, leave the PCB in one piece but you will need to adjust the measurements used for wire lengths and possibly the position of the DC sockets. You may need to use a larger case but this depends on your practical approach to 're-arranging' the set-up to suit a one-part PCB.

2. If you are having trouble getting polyester capacitors in either the correct value and or pitch it is perfectly acceptable to use ceramic or resin dipped capacitors. They may still not have the correct pitch but the longer legs they commonly come with will ensure minimal problems fitting them in place on the PCB.

Also of note is that the alternative diode (D2) is shown fitted on the PCB throughout. See the image below for how to correctly fit the preferred BYW80-200.



Attention (New Series) Dalek Supreme owners...

This next section effects you specifically. The unit built, as per this manual, is suitable for a standard 'two dome light' Dalek. If you have built or own a Supreme Dalek this requires specific changes to some components, wiring and PCB traces. This is because with the three dome lights of a Supreme the lights will draw 6A which exceeds the limits of some parts and areas of the unit. This is based on bulbs rated at 12V/24W which are regarded as the 'standard' item for the NSD builds. Before we move on to the specific changes here is a description of how you calculate the Amps the bulbs used will draw. The results are base on the bulbs being wired in parallel.

With a battery rated at 12V and using the 12V/24W bulbs you simply divide the bulbs wattage by the voltage the unit is powered by. In this case it should be 12VDC. So-

24W/12V = 2A

Simple isn't it? To clarify, dividing Watts by Voltage gives an answer in Amps. One further thing to do is multiply the answer by the number of bulbs you are using. So-

2A X 2bulbs = 4A (two dome light Dalek)

2A X 3bulbs = 6A (three dome light Supreme)

The Standard 12V/24W bulbs are the recommended maximum you should use. But you can use any bulbs you like of lesser wattage so long as they are 12V. The maths is done the same what ever the Watts of the bulbs. If your calculations for a smaller specification bulb come in at 4A or less then you don't need to worry about any of the following changes for the Supreme.

Changes:

There are four stages to making the unit safe to operate the increased load. These are-

Change the power switch.

Replace both DC sockets with an electrical terminal block.

Replace the internal and external power wires.

Increase the capacity of traces on the PCB.

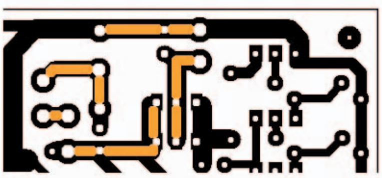
The suggested switch to use is from Rapid online and is rated at 10A the order code is- 75-0251.

This switch is large but is known to fit exactly where the 'standard' switch does with the parts specified in the parts list. You do need to make the mounting hole larger though.

The terminal block needs four sections so you have a +VE and -VE for both power in and lights out. The block should exceed 6A rating.

The internal wires and external wires should exceed 6A. Recommended wire is from Maplin's. It is figure-8 cable as before but is rated at 10A. The code is - XS71N

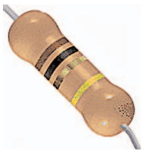
The final part involves the fitting of single core copper wire, as removed from household mains cable, to specific areas of the PCB traces. This is to prevent the burn out of traces when under the increased load. See the image for where the wire needs to be fitted. The orange lines denote the additional wire. The wire should be soldered at all joints to ensure good contact. The PCB pattern is reversed so you are seeing it as it looks from the correct view point.



A simpler and possibly more attractive method is to use a relay. The coil of which should be driven by the sound to light output. The relay is then used as a switch between battery and lights. The relay needs to be rated for 12V on the coil and the contacts must exceed 6A.

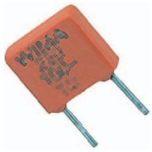
Component Identification

Resistors



The most common component in the circuit. The first three coloured rings on the body denote it's value. The fourth is it's tolerance, i.e. how close to the marked value it actually is.

Polyester Capacitors



Another common component. The values are marked on the components in digits with a suffix denoting their scale of measurement. This is most commonly uF or n. Ceramic capacitors can be marked slightly differently in that although they use digits they are a code rather than a direct value; i.e. a cap marked 104 is a 100nF cap.

Electrolytic capacitors

This component is similar in it's workings to the polyester type but it is constructed differently and has polarity, i.e. it is like a battery and needs to be connected the correct way around. The negative leg is denoted by the stripe that runs down the side of the case nearest the leg in question. They are generally in the uF range.

- Polarity Critical!

Diodes

Similar to the format of resistors there are two distinct types used and a third ‘alternative’ type. Two are glass bodied with a black end denoting the cathode. The third is a large square with a metal tab and has two legs that it ‘stands’ on. It's orientation is denoted by the letters K and A marked on the metal tab. The alternate given in the parts list is a very large cylinder with the same format as the smaller types illustrated.

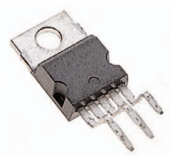
- Polarity Critical!

Transistors

There are 3 of these and although they look identical there are in fact two types. The internal st-ructure is different and you can only identify them by reading the identification printed on the face of the body. In this case you will find 2 marked BC548 and the remaining one is BC558. Again, these will only work when inserted the correct way around in a circuit.

- Polarity Critical!

TDA2003



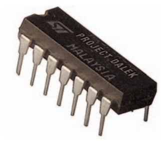
This is the heart of the audio output circuit. It is the amplifier chip and is easily recognised by the presence of it's five legs. It's identity is also printed on the front. It will only fit one way on the PCB and also requires that a heat sink is fitted as it heats up during use.

IRF540

This is a Mosfet. It is similar to a transistor but the internal structure is somewhat different. There are two of these and they are responsible for driving the filament bulbs.

- Polarity Critical!

IC's



There are four of these. The fourteen pin DIL IC is illustrated here. There are two others with 8 pins and one with 6 pins. They all have matching sockets. Although much hardier than they were in the early days of microelectronics, they must be handled carefully to avoid damage by exposure to static electricity. Leave them in their packaging until required. IC5 has a step down the length of it's body on the side that has pins 1, 2, and 3.

- Polarity Critical!

Potentiometer

There are three of these. They reduce or increase the amount of resistance in proportion to the amount they are turned. Volume, modulation rate and light sensitivity are controlled by these.

Work Space

One of the most important areas to deal with is the work space in which you intend to build the circuit. It should be clear of clutter and preferably in front of a window as daylight is preferable to build in. Failing that, a good light source like a halogen lamp will do at a pinch. The power leads to your tools, i.e. soldering iron, lamp, etc. should not present a trip hazard or be easily caught by a careless hand. Also, you don't want anything disturbed by family pets and/or children. Ventilation is important as the fumes from the soldering process can be injurious to health, in the long term and if anyone around you has a medical condition such as asthma. Always refer to the manufacturer's data sheets for full information. In the spirit of Health & Safety you should never leave a hot soldering iron unattended. Turn it off when you're likely to be away for more than five minutes. You are responsible for your own safety and that of others around you!

Tools

You need only a few basic tools to do this project these include...

- **A soldering iron** between 25-28W preferably with a pen type tip. It should also have a proper stand.
- **Small side snips** to cut and trim wires and component legs.
- **Wire strippers or a Stanley knife** to remove the outer sheathing of wires.
- **A selection of drill bits** from 0.5mm, 0.9mm, 1mm, 1.5mm and 3mm diameters for drilling any holes in the PCB that might be a little tight to fit components. For the case, you will need 1.5mm, 3mm, 6mm, 8mm, and 10mm HSS bits.
- **A heat shunt.** A light weight pair of tweezers that you can clip onto the leg of a component to prevent too much of the heat from soldering the joint getting to the components sensitive innards. A crocodile clip can also be used if small enough.

Helpful extras are a set of 'helping hands'. These are crocodile clips, on a stand that can hold the parts you are working on. Especially useful for holding potentiometers and the wires you want to join to them. One very useful item in my armoury is a piece of foam mouse pad with a fabric face but more on that later.

Solder

My one piece of advice on this is, for the novice, do not use lead free solder. Yes I know going green is important but it's a total pig to use. There is a knack to it and even I have trouble with it. Get traditional 60/40 Lead/Tin solder with a flux core. It should be 22-24swg in thickness for best results and ease of use. If, however, you are proficient with the new fangled lead free solder then post on to the Forum and tell us how to do it. If you are very new to the world of electronic construction I suggest you get some vero board and wire to practice the art of soldering, before moving on to the main project. Yes I know you don't want to do that, but do you want a unit that is most likely to work when you have finished, or not? You have been warned.

There are many instructional guides on the art of soldering and a quick Google will find a good selection. YouTube is also a good place to find guides. With the right iron and solder it is easier to master than you might think - and also quite relaxing.

Speed

Or the lack of, therein. Building the unit is not a race or at least it shouldn't be. If you rush things you will mess them up. Take your time building the kit. Prepare your components by identifying them, grouping them by values and type. This will pay dividends. The Assembly of the PCB is broken down into several parts.

The rule used here is: Lowest profile components first, in groups of the same value where applicable, and work your way through to the tallest. Do all this and you will find the build happens at a reasonable enough speed and it's most likely to work at the end.

Preparation

It is recommended that you clean the track side of the PCB prior to assembly. This is to remove oxidisation that may interfere with the soldering. Use warm soapy water with a kitchen sponge to gently scrub the PCB.

You can also preface this by using a fine steel wool to clean the copper first. Avoid touching the surface with your fingers as much as possible, as the natural oils from your hands may interfere with the soldering.

Before you even touch your iron, lets prepare ourselves, our tools and the components. Is your work space clean tidy and ready? Do you have all the tools you will need? Right then lets prepare the components...

You need four pieces of either bell wire or off cuts from component legs to make the wire links. These can be lined up on the PCB so you can gauge where to bend them. Set them aside once done. By the way, bends on the wires and any other components should not be sharp and preferably use round nose snipes or, in the case of the resistors, do what I did... from an old computer motherboard I removed the plastic surround from an IDE header. At one end I cut a notch each side, in the walls of the plastic. A resistor fits perfectly inside the width of the header and the component legs sit in the cut notches. Now, simply, with your finger on one leg and thumb on the other, push down gently. You now have a resistor that has it's legs bent to the correct pitch, with nice radii, ready to drop into position on the PCB.

This will also work for the diodes, by the way. If you don't have an old motherboard to rob for this simple and useful gadget you can make it from anything. Your only guides are it should be no less than 9.5mm and no more than 10mm wide.

Now refer to the sequence of assembly below and in their groups of value, bend all the diodes and resistors so they are ready for quick insertion.

Be methodical. Do the greater quantities within each group first and work through. Don't mix them up. If you do, you can reference the codes printed on the diodes against the list to identify them. The values and colour codes are also noted for the resistors.

Get all the components identified and laid out so they are easy to locate as you progress down the list. You will notice that within some of the component groups there are smaller subgroups. Take a look, for example, under the resistors heading. There are four groups. The first has four 1K resistors, the second is made up of four 100K resistors. The third is a group of three followed by yet another group of three.

The next is four groups of two. You can see where this is going can't you? Follow the list; insert a group of components, solder, trim ...and your ready for the next group. Select, insert, solder and trim ...and so on ...and so on. The idea is not to fit so many components in one go that you are fighting through a forest of legs to solder them. It also provides convenient 'break points' so you can stop and know where to pick up again.

You will also notice that as mentioned before, you are also working from the lowest profile component up to the tallest profile component. Trust me, it's a good method to use.

Heat: *the friendly enemy*

OK. Here's the thing: we need heat and a fair amount, to heat the intended joints and the solder to form a good joint. But on the other hand some components are particularly sensitive to heat and if you’re careless you will fry the innards. Also if you’re not careful, you could de laminate the solder pads from the PCB, which really sucks.

Basically, this is one good reason to practice if you’re new to the hobby. To assist with the likes of the transistors, diodes, TDA2003 and the Mosfets you can use a heat shunt on each leg as you solder the joints to prevent damage to the item. At the end of the day it’s all about practice, practice, practice.

I can solder such items without recourse to the shunt but I can see if things are going right or wrong quickly enough to back off. When I started this hobby I messed up a few boards and learned the hard way. You don't have to... honest.

My little helper...

One of the most useful things I have in my tool kit is a piece of mouse mat cut slightly larger than the PCB. The mouse mat itself is the rubber sponge type with a cloth face. It provides a good anti-slip surface to rest the PCB on, when soldering up. The other benefit is that once the board, with components, is laying on it, track side up, everything is held up against the PCB. You will find that at times if the group of components you are about to solder are at one end of the board, it will tip slightly. To get the board so it sits flat, put a chock of some description under it. I also use my helping hands to press the board down and level, especially as the component heights increase. So load your PCB with a group of components, put the mouse mat on top (cloth face down) and turn the board over. Level if required either by supporting in the right place or use a set of helping hands to assist.

Alternatively you can 'splay' the legs of the components to hold them in place, but, I prefer to leave the legs straight as it's easier to remove the components if you need to de-solder for any reason.

Before you start...

Boxing clever...

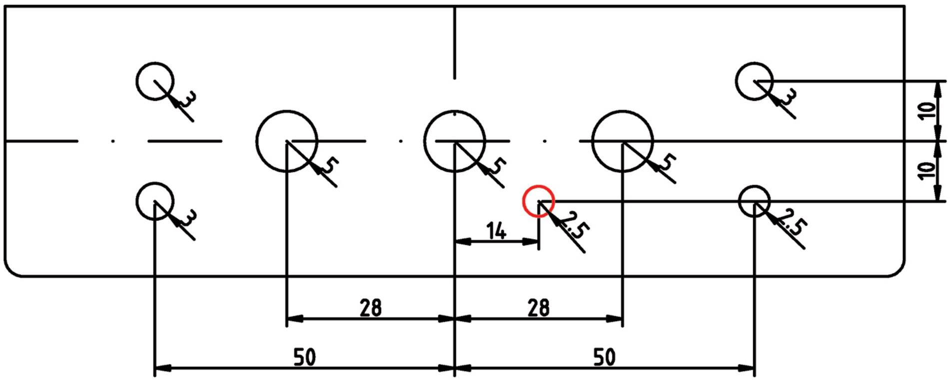
Another thing to do before you begin assembly of the unit is to prepare the box you intend to use to house the unit. If you are using the box as listed in the parts, then you can refer to the diagram for positioning and for the diameters of the holes you need to mount everything. In particular, you need to prepare the box with reference to the mount points that will hold the PCB in place. Should you choose to use your own choice of case, you should prepare the box similarly, once you have worked out how you intend to arrange the box mounted parts.

OK, you have read this manual back to front, practiced soldering if needs be, have prepared everything and are ready to roll. The next section guides you through each stage. At the end of which, there is a picture showing the PCB with all the components you should have installed at that point.

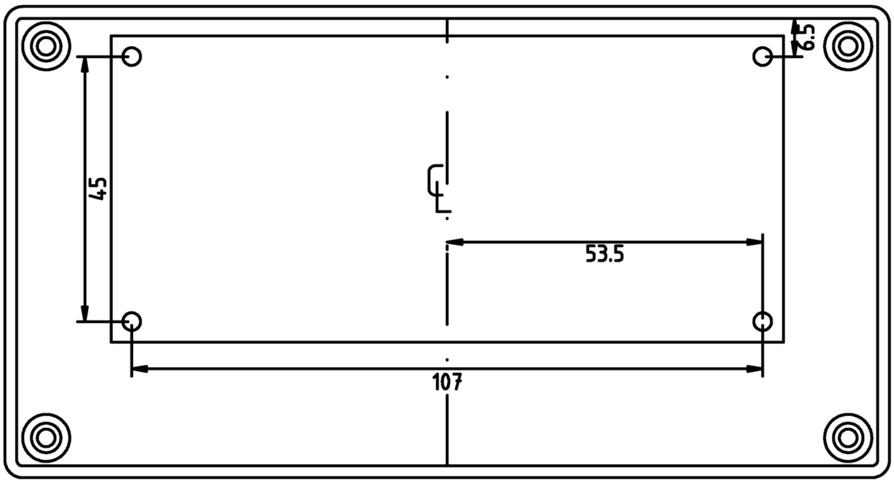
Good luck, have fun...

Box Details

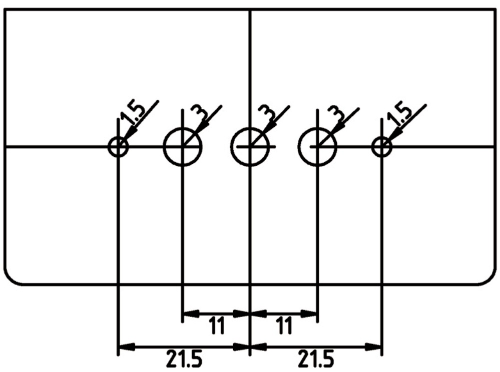
The dimensions given suit the case as listed under parts. You may at your own discretion alter the layout to suit yourself.



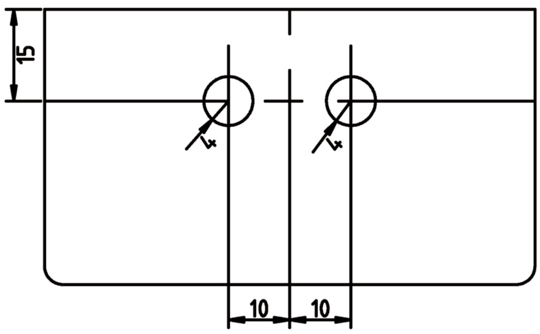
Front panel: showing the placement of the controls and audio I/O.



Plan view of box: showing mounting holes for the PCB, bottom of the picture is the front face of the box.



Left side, in relation to box front.



Right side, in relation to box front.

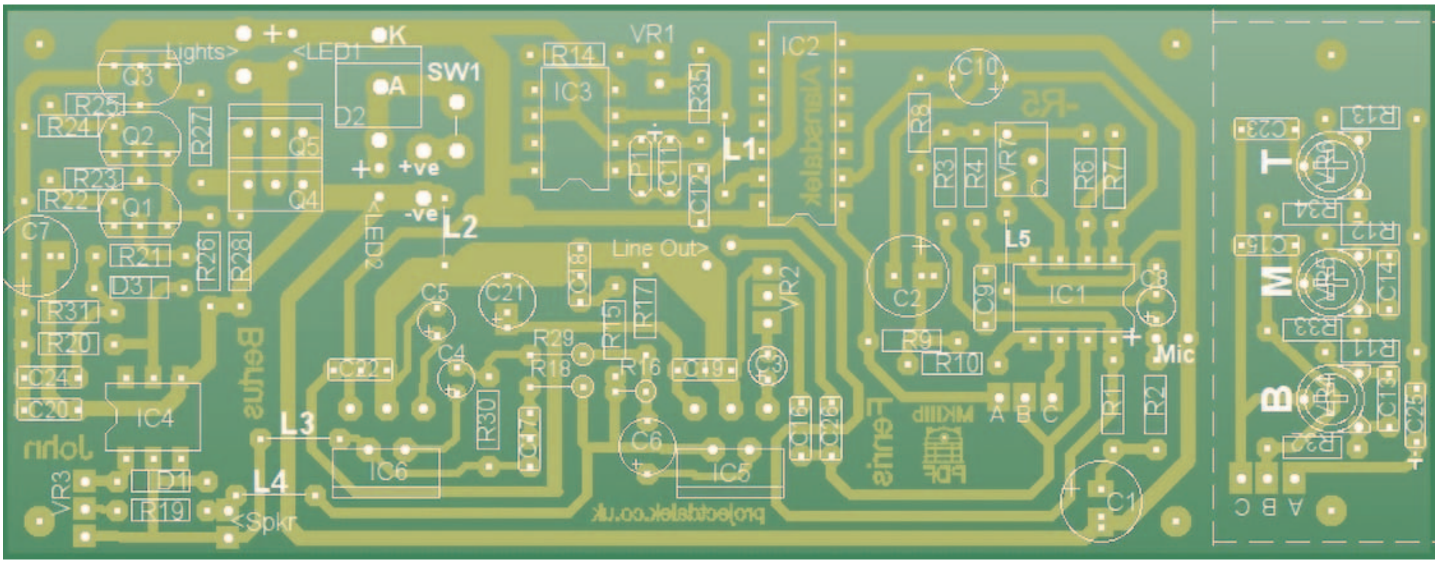
The holes can be drilled accurately by using the bare PCB as a template to mark the holes prior to drilling. These should be 3mm in diameter. I used M3 12mm, long cheese head pozi screws in this sequence;

Screw with washer through the bottom of the box, then another washer followed by two nuts. This gives 4.5mm clearance under the PCB. If you use countersunk M3 screws you can omit the first washer. Remember to counter sink the four holes.

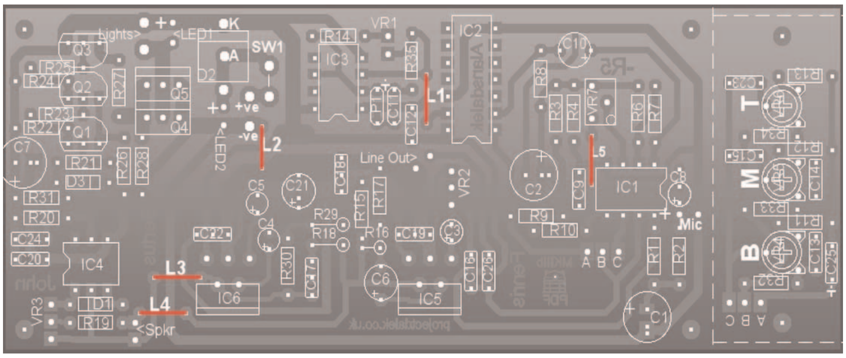
M3 9mm bolts will also work fine for fitting. There are other options you can use to mount the PCB including mount pillars (as used for PC motherboards) and self adhesive nylon pillars, although the feet of the back two may require trimming to fit. For marking up the mounting holes and then drilling them, it is suggested you use marking tape to cover the faces. This will assist in drilling without fear of slipping and scratching the case.

Sequence Of Assembly

Bare PCB.

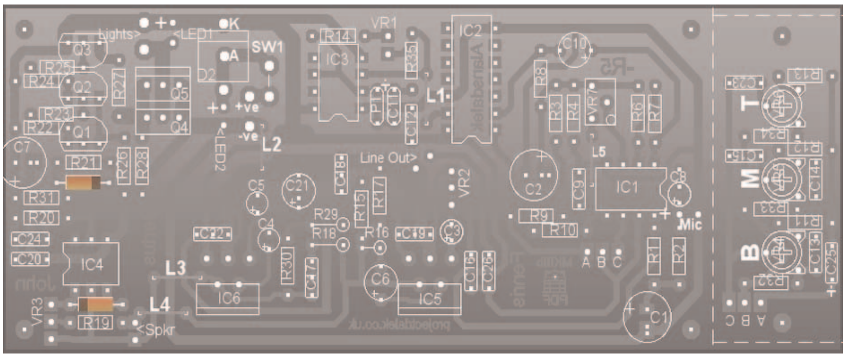


Step 1: Wire Links



Wire Links L1, 2, 3, 4 and 5

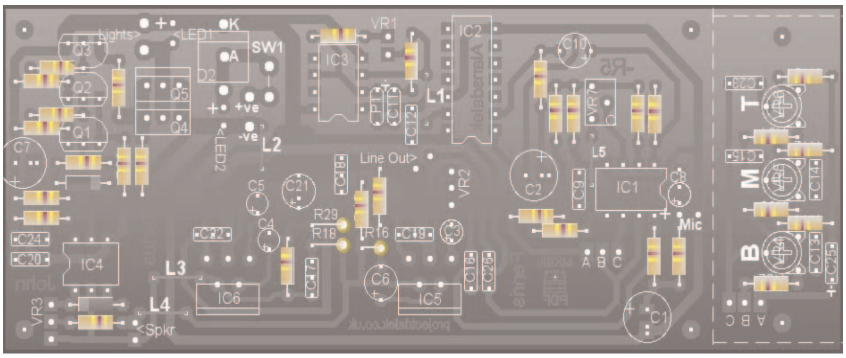
Step 2: Diodes



D1, D3
1N4148

NB. Remember to pay special attention to the orientation (polarity) of the Diodes.

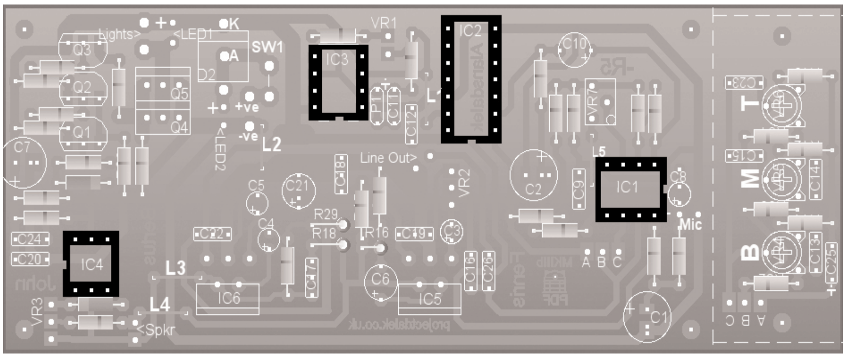
Step 3: Resistors



R1, 21, 22, 24,	1K	Br, Blk, R
R2, 23, 25	10K	Br, Blk, O
R7, 10, 20, 31	100K	Br, Blk, Y
R19, 26, 27	100R	Br, Blk, Br
R3, 4	220K	R, R, Y
R8, 9	470K	Y, V, Y
~~R5~~	not used, do not fit.	
R11, 32	12K	Br, R, O
R12, 33	3K9	O, W, R
R13, 34	1.8K	Br, Gry, R
R15, 18*	1R	Br, Blk, Gld
R17, 29*	10R	Br, Blk, Blk
R6	33K	O, O, O
R14	8K2	Gry, R, R
R16*	220R	R, R, Br
R28	2.2K	R, R, R
R30	470R	Y, V, Br
R35	82K	Gry, R, O

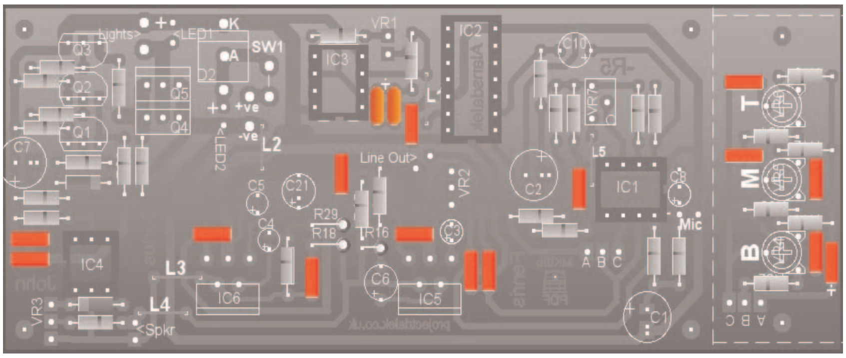
* Fit these three resistors last as they are 'on end'.

Step 4: IC Sockets



IC 1, 2, 3 & 4

Step 5: Polyester/Ceramic Capacitors

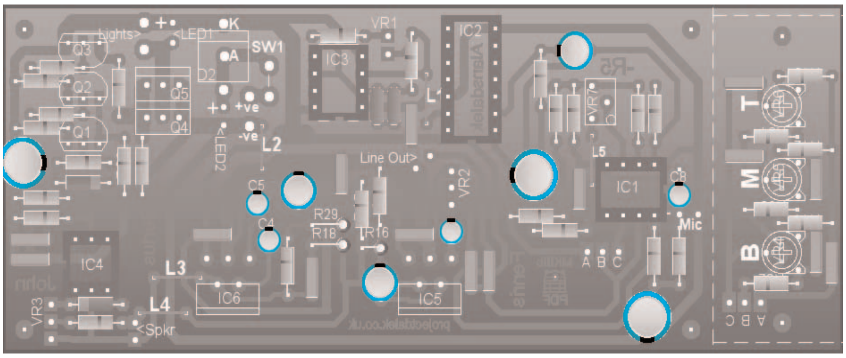


C9, 12, 17, 18, 19, 20, 22, 24, 26*	100nF
C11, P1**	1uF
C14, 23	4.7nF
C15, 16	22nF
C13	47nF
C25	1uF

*Fit C26 only if you are fitting the line out.

**See preface for important information on component options.

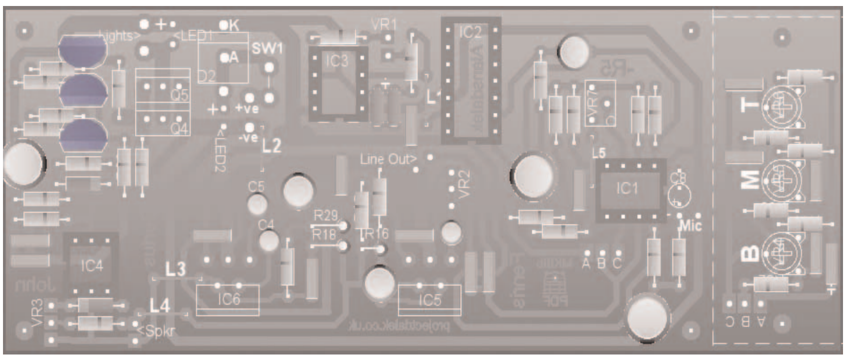
Step 6: Electrolytic Capacitors



C1	33uF
C2, 4	10uF
C3, 5	4.7uF
C6	22uF
C8, 10	2.2uf
C7	470uF
C21	220uF

Polarity is critical when fitting the capacitors. Please note the position of the band, on the side, during assembly.

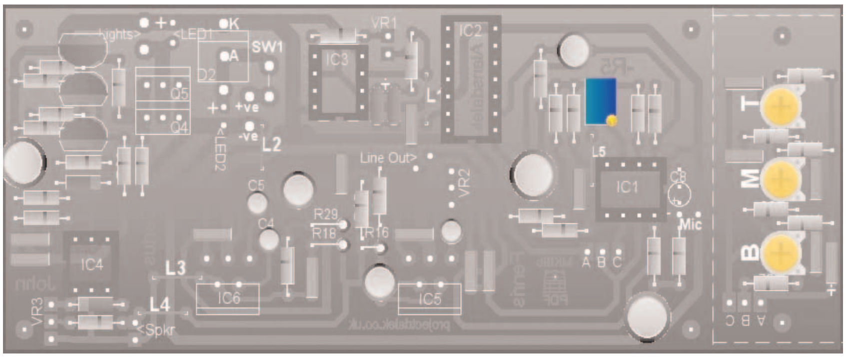
Step 7: Transistors



Q1,2	BC548
Q3	BC558

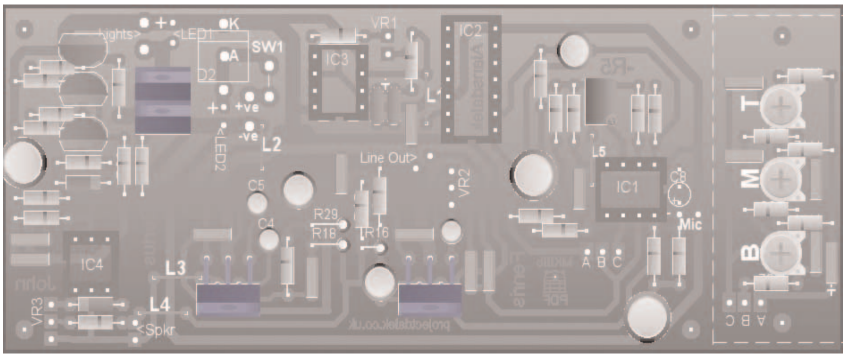
NB. Pay special attention to the orientation of the transistors.

Step 8: Variable Resistors



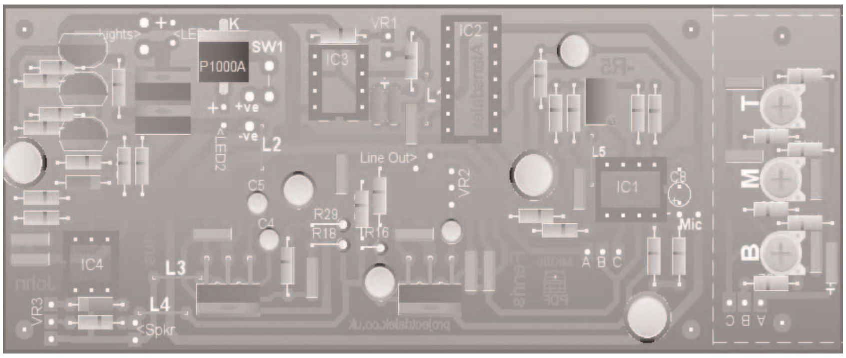
VR4, 5	100K Cermet
VR6	500K Cermet
VR7	500K Bourns

Step 9: Amplifiers and Mosfets



IC5, 6	TDA2003
Q4, 5	IRF540

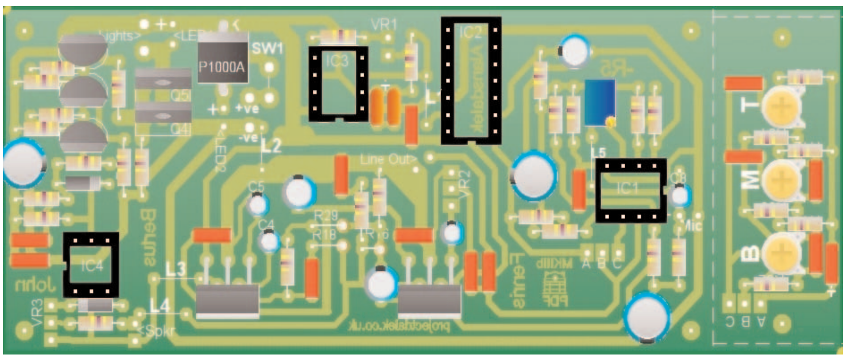
Step 10: Diode



D2	P1000A

The preferred Diode is as listed in the parts list. Note. The diode listed, shown, is the alternate of the two recommended. See preface.

The PCB at this stage is fully populated. All that is left to do is prepare the wires that will connect the off board components to the PCB.



Care must be taken with connecting the wires to the peripheral items.

The variable resistors must be wired as per the diagrams. The same goes for the audio and power connectors. Follow the guides carefully. In the event that your cables have different colours than those illustrated, note, on the drawings, your alternate colours. Particular attention should be paid to the power in socket and the audio in/out sockets.

Wires and Wiring

There are three types of wire used for connecting the unit to it's peripherals. These are: Hook up wire, Servo cable Figure 8 power cable and audio cable. See parts list for order numbers and distributors. Hook up wire: This is used to wire either or both LED's to the PCB. Length should be 50mm minimum which should be more than enough to fit them to the PCB and the fascia. Two colours of wire are suggested as a reference for polarity. JR servo cable: This is a ribbon cable consisting of only three wires. Each wire is coloured for identification. They are orange, red and brown. The colours are used on the wiring chart further on in the manual. Cut three lengths to the following.

For VR1 cut a 70mm length and remove the brown wire.
For VR2 cut an 80mm length For VR3 cut a 110mm length

Remove 5mm of sheathing from one end of all the wires. Twist and solder each wire then fit to the PCB following the colour chart. Other servo cables can be used but note the colours may be different.

Auto wire also known as figure 8 wire: Cut a 90mm length for the power in. Use with the 2.1mm DC socket. Cut an 80mm length for the power to lights. Use with the 2.5mm DC socket. Cut a 110mm length for the switch. Audio cable: Audio in (mic) 70mm Audio out (speaker) 160mm

DC sockets

There are two of these. Their construction and pin layout are identical but they are of differing internal pin diameters. The reason for this is to prevent incorrectly plugging the power into the light socket.

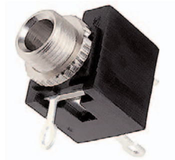
It is suggested you use the smaller diameter socket for the 12V DC IN. The wire to use should be rated at no less than 6A. I used 10A rated figure of eight cable, as used for car wiring, for all the power related items. It is also coloured coded red and black. Strip 6mm of outer sheathing from all ends of the wires, twist and tin the ends. Tin the two terminals as indicated and solder the wires to them.

Switch

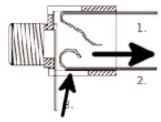
This should be rated at no less than 5A and preferably it should be higher. Some switches come with three terminals and others with only two. In the case of the three terminal type, use the centre and one other to connect the wires to.

Again use the figure of eight cable. Strip 5mm of sheathing and prepare the terminals by tinning and then solder the wires in place on the switch.

Audio Sockets (Speaker & Mic)



Note. These are stereo sockets. There are 2 of these to wire up. You will need two lengths of single core audio cable cut to the lengths specified in the 'Wires and Wiring' section. Remove 25mm of outer sheathing from one end of both twist the shield wires and tin. Cut 5mm of the sheathing from the core wire, twist, and tin. The other end of each cable should have 15mm of the outer sheathing removed and 5mm off the inner core. Solder the wires as before. One modification to the sockets before wiring up is the removal of terminal 2. See diagram. Behind terminal 3 is a small slot. With a small screwdriver press the metal surface visible through the slot up and towards the back of the socket. Terminal 2 should slide out with the attached contact plate. Discard..



LED

This little component is your visual reference for what the dome lights are doing. It has to be wired up correctly with reference to polarity. To start you need two lengths of multi-strand hook up wire, preferably of contrasting colours. Red for +VE and white for -VE, perhaps. Strip 6mm of sheathing from both, twist and tin.

You will notice that the LED legs are different lengths. The shortest is the negative (pin 1. in the diagram) also accompanied by a flat edge on the LED's flange. Tin the leads but be fairly quick as this device is sensitive to heat, use a heat shunt for extra safety if you're unsure. Slip 1.6mm heat shrink over the joints once both wires are attached. I tend to use long enough pieces to cover the whole leg as well.

Now twist the the wires into a single pair holding the wires at the component end so you don't damage the legs. Note what colour wires are attached to which leg for later. It is suggested you shorten the component legs by half, at the most, for ease of fitting.

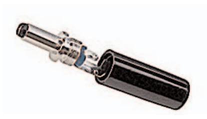
- Polarity Critical!

Power plugs

Plugs: the Red denotes the positive terminal to which the red wire should be attached and the black where the black wire should be attached in both instances. This is critical so ensure correct wiring to avoid disappointment

There are two of these and they should be different internal diameters to match the DC sockets. The smaller of the two is for the 12V DC IN. Whether you use 6A or 10A figure of eight wire you will find that the anti strain tangs at the top of the long terminal will only wrap around one of wires of the pair.

The picture (previous page) has been coloured to show how the terminals relate to the pins. The shorter of the two terminals is the positive, which is the centre of the pin. The longer terminal is the negative which is the long shaft of the pin. Strip 6mm of sheathing from the black wire and the red wire. Twist and tin the ends.



Slip the body in place on the wire the correct way around. Now you need to present the wire ends to the terminals to see how much of the exposed wire you need to trim back. Your guide here is that the black sheathing must be positioned so that the strain relief tangs, when crimped, grip the sheathing. Its wire must not be so long that it goes down the centre of the plug and short the positive contact. You can then adjust the red wire to suit. Both wires should be soldered on the inside faces of the terminals. Once soldered crimp the strain relief tangs around the black wire. Slide the body down the wire to the plug whilst holding the wires.

Once pushed all the way down and still gripping the wires close behind the body, screw it down into place. Hopefully if you have bored enough clearance into the body, the internal structure won't be twisted by the screwing motion. It's a good idea to check (using a multimeter) that nothing untoward has happened. Do the same for the light jack plug.

Alternative DC Power I/O

A 'lo-tec' but equally valid solution to connecting the, 12V DC IN and the lights, (to the unit), is to use electrical terminal blocks. You will need to drill a hole in the case to bring the power and light lines out from the box to the terminal block which should be bolted to the case. Secure the wires and label the connections to avoid incorrectly connecting the external power and the lights.

At this stage you have completed the primary build: the PCB and the secondary: the off board components. You should now take a moment or two to visually check the soldering for any faults. Dry joints, bridged tracks, pads. Check the orientation of all those components that need to be orientated correctly in order to work. Check the wiring of the off board parts. Remember, if your wires were different colours from the diagrams, you should have made notes on those differences to aid in the correct fitting of these parts to the PCB. This is critical! A mistake here will render the unit inoperable and at worst, if the power is wired incorrectly, permanent, irreparable damage may occur. Once you are happy that all is well, it is time to move on to the joining of the the off board parts to the PCB

Follow this sequence...

Light socket, Power socket, Switch.
VR1, 2, and 3 wires
LED1 and LED2 (if fitted)
Mic, Speaker

Connecting the Off Board Components to the PCB

KEY:

DC Sockets & SW1

Red = +VE

Black = -VE

LED1 & LED2

Red = +VE

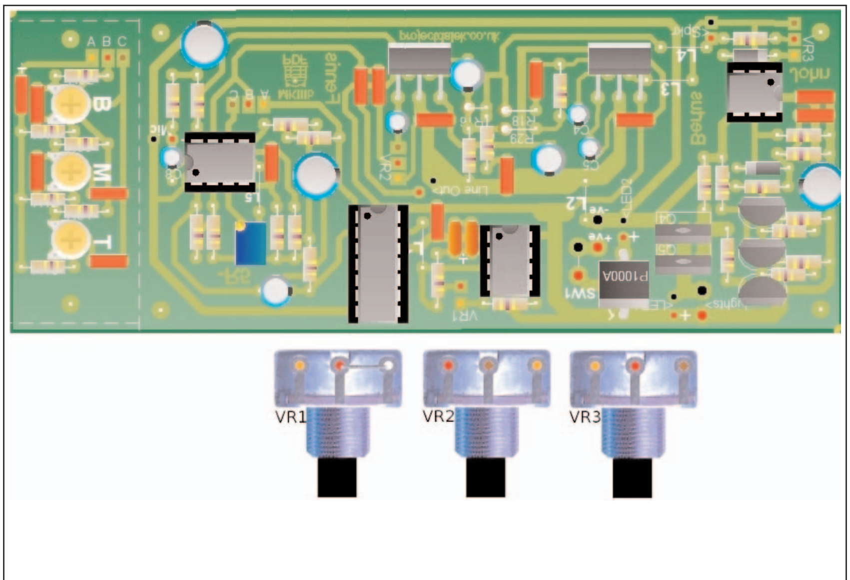
Black = -VE

Mic & Speaker

Red = Is the core wire. For the mic the correct header hole has a + next to it

Black = Is the shield wire

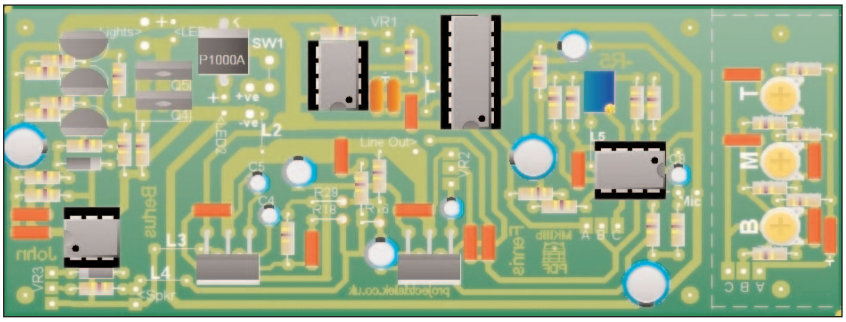
VR1, 2 & 3 = Wire as per colour code. Remember this is JR servo cable colours. If yours differs, make note of the changes, on the picture to aid you.



In the event that you had audio cables or wires which used differing colours for the wires, refer to the variable resistor wiring diagram and any notes you have made as suggested in the variable resistor wiring instructions.

If you used different coloured cables to wire your unit up, make a reference to guide you through this last bit. Also remember VR1 and 3 are the 10K Linear potentiometers. They will be marked: B10K. VR2 is the 10K Logarithmic potentiometer and will be marked: A10K.

If you have made the capacitor switch to take advantage of the extra range this gives to the modulation, you may wonder where to put the switch. In part, this will depend on its physical size, so simply ensure that wherever you put it, nothing will touch the terminals on the back of the switch. A sub miniature toggle switch, with the cap attached, can be fitted between the potentiometer with the mounting hole level with the two uppermost 6mm mounting holes.



Well, that's all the hard work done. There's just a couple of jobs to do before you start fitting the unit into its case: fitting the 4 DIL IC's and the heat sink on the TDA2003. First off, place the PCB on a non conductive surface, the mouse mat would be good, near a radiator, kitchen sink, or central heating pipes. The reason for the proximity to metal work is to use it as an earth point before you handle the IC's. Historically IC's in the early days of the technology where fragile in the presence of static electricity. Basically if handled incorrectly the static charge from the human body could destroy the IC. These days they are not so susceptible but old habits die hard and I'd rather not take the chance. Having touched a bare metal surface you can handle the chips, one by one, ensure they are orientated correctly referring to the PCB diagram if required. You will most likely find that the legs are too splayed to fit in the socket so very gently, with your fingers, adjust them until they all line up with the socket's holes. Once aligned, gently push down until fully depressed.

Fitting the Heat Sinks

The heat sinks are simple to fit. Firstly you need to remove the pins. Those supplied in the kit will have press fitted pins. These should be removed by using small pliers. Grip the pin and twist back on itself and it will come away from the sink. The raised square on both sinks should be filed down flat. The sink for IC5 needs no other work done to it for fitting. The sink for IC6, however, does need a little work for it to fit in place correctly. Hold the sink up to IC6 and note the wire links L3 and L4. As you can see the sink cannot sit down correctly because it is sitting on the links. To remedy this simply file two notches in the relevant places on the sink so that the wires and sink do not touch when the sink is in place. The sink may need a little adjustment where it sites between the capacitor and resistor on the opposite side to the links. Next, using an M3 x 9mm bolt with the supplied bush to fit the sink in place, place the bush on the bolt, thread this through the front of the chip’s metal tab. Now on the back of the chip, place the supplied silicone thermal pad on the bolt, followed by the heat sink and lastly the nut. Tighten up more than finger tight but not so tight as to distort the metal tab or sink which will effect the thermal transfer between chip and sink. NOTE. The bush will need adjusting slightly so that it fits flush with the back of the chips metal tab when in place.

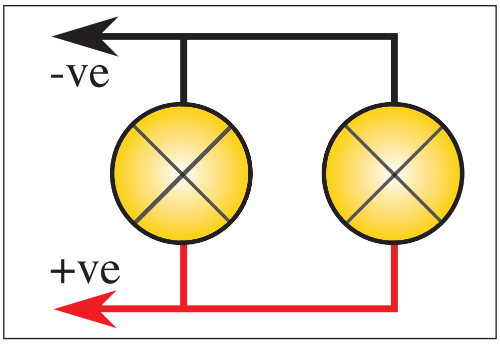
Fitting the PCB in the Case

This is the suggested method of installing the PCB into the case. First up, the PCB should be placed into the case with the heat sinks on the opposite side of the case to the mounting holes for the pots, switch etc. Once in, fit the nuts to secure it. Now there is one slight variation to deal with before proceeding. The PCB has the option to fit a second LED. LED1 is for indicating what the dome lights are doing. LED2 (the second LED) is a 'power on' indicator. If you have only fitted the dome light LED this goes in the lower right hole on the front of the case. If you have fitted the second LED, LED2, then this should be fitted in the lower right hole. This would place it under the power switch when that is fitted. The dome light LED, LED1, will then need to be fitted elsewhere. It is suggested that this LED is fitted between the volume pot and the light sensitivity pot level with the power light hole. With that covered and the PCB in place. Fit the LED or LED's into the respective holes. Now fit the variable resistors in place. It is a good idea if the shafts have been cut down to between 8 or 10mm. This should be VR1 10K Lin in the right hole, VR2 10K Log in the middle, VR3 10K Lin in the right hole. Now fit the switch in the upper right hole, The speaker socket should be fitted in the lower left hole. Route its cable around the back of the heat sinks. Fit the mic socket in the upper right hole. Now fit the ‘power in socket in the right hand hole in the end of the case and the Light socket in the remaining left hand hole. The Tone control panel should be mounted with 3mm bolts and spacers on the other end of the case. Orientation should be with the wires on the upper left side of the tone control PCB. This orders the controls, from left to right, Bass, Mid, and Treble.

The Dome Lights

These require wiring in a specific manner. The sound to light can handle up to a single 12V/50W bulb. We will however be using two 12V/24W bulbs. Anything less than 24W is also fine so long as the bulbs are 12V. These should be wired in parallel, not series. See schematic (right) for guidance. Polarity is not an issue, one line goes to positive, the other to the negative.

Here is a rule of thumb to aid selection of a fuse of suitable value. The unit as is with the 12V/24W bulbs wired in parallel will draw at peak 4A. You need a fuse rated at 1.25x the Amps drawn. In this case that would mean a fuse rated at 5A. Using bulbs of lower wattage will reduce the amps drawn as will using LEDs so you must measure the Amps your unit draws, if you use alternative types of lighting, to ensure a fuse of the correct value is chosen. In one particular instance a higher value fuse may be required if using the ‘standard’ bulb type. That is if you have a Supreme NSD. This has 3 dome lights and will draw 6A with the correct bulbs fitted which means a 7.5A fuse is required. If you use standard LED types you must calculate the value of the series resistor each one needs. Car LED ‘bulbs’ can be used without this worry.



The Controls

Light Sensitivity:

This controls how sensitive the lights are to the input. Fully clockwise is full sensitivity. Fully anti-clockwise is 'off'. The lights are dependent upon the amount the volume is turned up or down. At maximum volume you would need to decrease the light sensitivity by turning the control anti-clockwise until the lights react as you want. If the volume is low you would have to turn the light control clockwise to get the reaction you require. Notice the LED on the front panel flashes in response to give you a sense of what's happening with the dome lights when operating the Dalek.

Volume:

Does exactly as it says on the tin. Anti-clockwise decreases the volume and clockwise increases the volume. With careful set-up, you should achieve full volume. This is dependent on following the guide with reference to speaker(s), mic, gain and EQ settings.

Modulation Rate:

This controls the effect of the unit on your voice. Clockwise will increase the 'chop' effect, anti-clockwise decreases the effect. The unit has been 'tuned' to be able to effect your voice within the relevant frequency's of most Dalek variants since the beginning of the series. Please see the addendum for variations on set up of this facility and a guide to dial position and Dalek voice types.

When setting up the unit, if you have a boom mic, you can plug in the mic and ear piece to test the unit and tune it ready for trying out the speaker(s). Once you are happy you can unplug the ear piece and plug in the speaker.

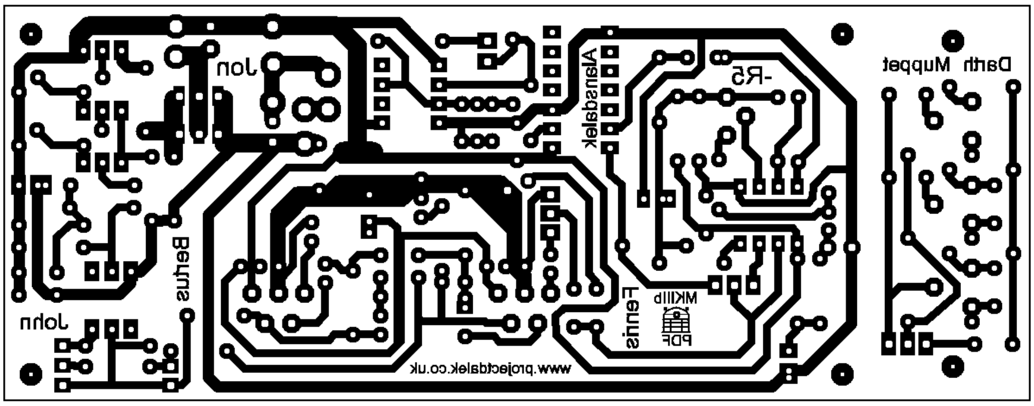
Optional modifications:

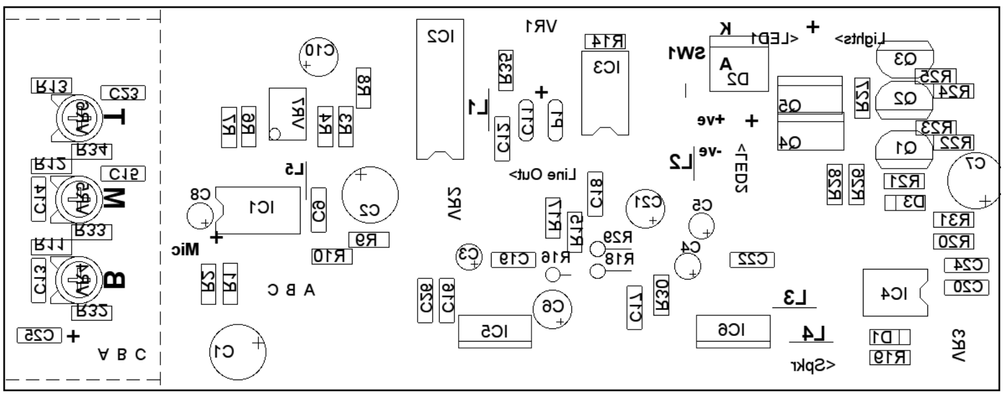
Provision has also been made to fit a line out socket should you wish to bypass the built in amplifier. This will require the fitting of a 100nF polyester or ceramic capacitor with a 5mm pitch. You will also require a suitable socket and a piece of single core audio cable. Remember the core wire is the Signal and should always contact the tip of the plug that is plugged in from the external amp. Note that the internal amp will still function. It has to, in order for the lights to work.

Although the sound to light part of the unit is geared towards driving filament bulbs it can drive LED's so long as they are connected correctly with regards to polarity. You will also have to take into account the voltage present and how it varies in response to the audio input. Correct value ‘in line’ resistors will need to be fitted to prevent burning out the LED's. Alternatively source 12V LED's with a built in resistor.

PCB Artwork

Produced here, is the artwork for the solder side of the PCB and the silk screen. If you wish to produce your own PCB you will need to ensure the measurements physically fit the given sizes. You do not need to reverse the image when transferring to the copper side of the PCB. The artwork for the component side may also need scaling to achieve the same measurements and is reversed already. There are a few methods for toner transfer of circuits. Some involve expensive specialist papers for use with laser printers. Other methods involve the use of matt or glossy photo paper, again, with a laser printer. If this is of interest to you then see my guide on the following pages. On a personal note, I had the best results with photo paper and it works out far cheaper, per sheet, than the supposedly dedicated transfer medium. There are also a selection of video's on YouTube that demonstrate these methods. See the links in my Voice Mod topics, on the Project Dalek Forum.





Kit Options

Option 1: A ready made PCB with silk screen printing is available if you prefer.

Option 2: A kit of parts including the following: PCB, all on board components, variable resistors, LED and the heat sink, (further items will be required to finish).

Option 3: A 'full house' system is available. This will be a complete, boxed, unit. All you have to do is plug in your headset, power and lights and you're good to go, (the plugs for power and light will be supplied for you to wire up).

Explicitly for any of the above, contact Fenris via private message on the Project Dalek Forum.

All technical queries will only be dealt with via the Project Dalek Forum (see the topic related to this unit). This will ensure that the questions and answers will create a useful knowledge base, for all members to use.

Additional Instructions for scratch built PCB

- **Fine steel wool.**
  This is to breakdown and clean any oxidisation and natural oils from your hands from the surface of the copper.
- **A hacksaw or junior hacksaw.**
  This is for cutting the PCB to size. Always cut with the copper face upper most.
- **Emery paper or similar.**
  This is to remove the burrs from the cut edges of the cut PCB. Do this without fail before transferring the PCB image.
- **Single sided copper clad board.**
  This is available in small medium and large sizes. It is suggested that you get an A4 size piece so you have spare in the event of failure. Note that you should not use 'photo sensitive' PCB
- **Ferric chloride crystals, or sodium persulphate.**
  This is for etching the PCB. They are supplied as crystals which should be mixed as per the instructions. Ferric chloride can also be bought as a ready mixed solution in small quantity's. Observe all health and safety labels without exception.
- **Etch resist pen.**
  This is as it says resists the etching process. Useful for repairing tracks or pads that have not fixed properly to the PCB during the toner transfer process.
- **Plastic tray.**
  This is used to hold the etching fluid and etch the PCB in.
- **Plastic tweezers.**
  Used to handle the PCB during etching.
- **Rubber gloves, safety glasses** and old clothes should be used/worn.
- **It is possible to buy a etching kit** that will include everything you need bar the PCB. You may find that some are also supplied with gloves and safety glasses.

Note that no metal tooling should be used during the etching process.

My first attempts at etching were done using 'press n peel'. The results were not too disastrous but were unnecessarily expensive and in the end not reliable enough. I wont blame the media entirely but it was just to damn finicky, and at £15 for five letter size sheets, too costly to keep fighting with. After that, a different media was found that everyone else, in the know seems to use..

Photo paper is the answer. Either gloss or matt can be used and there's no need to use expensive branded types. The only pre-requisite is that it should be compatible with laser printers. That said what I used was aimed at ink jet printers and worked fine. In fact it proved itself to be a very worthy material: stable, 99% first time correct and less fraught with doubt. With this sudden learning curve under my belt and success after success, in such a short period, I put together this guide to making your own PCB's, which first appeared on the Project Dalek Forum and is reproduced below...

Safety First

Rubber gloves and safety glasses are a must. You'll need a dust mask for when you saw and drill the PCB. Ferric chloride stains all it touches and it's not a good idea to be near any metal work such as the kitchen sink. Please ensure yours and others safety during the following processes. It is YOUR responsibility. Read all safety labels on the chemicals used and the manufacturers data sheets.

To begin with, you will need...

- A pack of photo paper as mentioned above.
- An iron, preferably one that isn't currently being used for housework.
- An A4 envelope, cut up so you have one side of it (discard the rest). Ensure you have removed any sticky parts if they are self sealing and the flap if it's a lick 'n' stick.
- A sheet of PCB board, single sided, it should not be photo sensitive PCB.
- A firm surface to iron on. I use a piece MDF.
- A hobby drill preferably with a drill stand and a selection of drill bits. Common sizes for electronics are 0.51mm, 0.9mm, 1mm, 1.5mm, and 3mm. They can be HSS bits or you can buy PCB bits. The PCB bits are more expensive but are far superior.
- A bowl of water (warm).
- Sand paper and fine steel wool.
- An etch resist pen.
- Scissors.
- Ferric chloride or other PCB etchant.

A Brief Overview Of The Process

Laser printers use toner cartridges. This toner is a type of plastic and during the printing process, it is fixed to the paper by heat. Simple isn't it? Now the clever thing is that we can use the heat from a household iron to transfer the image on the paper to the copper clad board, basically, the toner melts slightly and sticks to the copper.

Note that the image you have from a magazine will be correctly orientated. Or having created your pattern on the PC on the bottom of the PCB in software it should also correctly orientated so that after transfer, if you could see through the component side of the board the pattern would look as it did in the software. Print it off on photo paper without reversing the image. Only copper traces that are created on the top side of a PCB would need to be reversed for printing.

Lets Begin

First up you will need the PCB pattern. Either it is the one supplied within this booklet or if you are working on a different project, it could be from an electronics magazine, a website, or one you created yourself on a computer, with the appropriate software.

Your printer should have a relatively high DPI and be a laser printer. Now it is possible that your printer may not be able to deliver a dense enough image for a good quality transfer that will resist the etching process.

You will have to try first to see if your printer can produce the desired density. If you do not own a laser printer then all is not lost. I don't have a printer but I am lucky enough to have the PCB software that fits and works on my USB stick. I am also lucky that the local printer is OK with plugging said USB stick into their PC's to print off the pattern on their industrial sized printer. You can, if you have an ink jet printer, print off your pattern and then have it photo copied onto the photo paper at the printers. Ask them to ensure the density is set high on the copier.

My PCB's have been no larger than 150mm x 67mm. Beyond this size I have no experience of any problems that may or may not occur when trying to do larger PCB's using this method.

Preparation

Now you have your pattern, it is time to prepare your copper clad board. Cut out your printed pattern and use it to mark out the board. Try to avoid touching the pattern surface. You should aim for the piece of PCB to be 5mm oversize all round minimum. Use a hack saw to cut out the piece, with the copper face up. Once done, you should sand the edges to de-burr them.

Now you will need the fine steel wool. Using a small circular motion you should polish the copper face of the board. This is to remove any oxidisation and natural oils passed from your fingers onto the surface. In warm soapy water clean the board, then with a lint free cloth, dry it thoroughly and do not touch the copper face again. Prepare a bowl of hot water ready for the following stage.

Transfer

Iron switched on? Firm surface ready? Good.

The iron should be set to maximum heat. If it is a steam iron ensure that the steam is turned off. Now lay the PCB copper side up. Place the piece of envelope on top of the PCB. Place the iron on top and press firmly down onto the board. Move the iron around to ensure that all the copper is heated thoroughly. On my PCB's this was done for around a minute. Once warmed, remove the paper, get your pattern and place it, face down, on the copper. You will notice the board is very hot, so be careful. This is the reason why the board is oversize. It is easier to place the pattern roughly in place rather than trying to line up two identically sized objects when one will burn your fingers.

Briefly, at each end or all four corners, press the pattern down then quickly put the envelope paper back on top, followed by the iron. Press down firmly for a count of ten. Inch by inch move the iron around the board pressing firmly at each stop and count, again, to ten. Pay particular attention to all the edges. Holding the envelope paper with one hand you can move the iron around easily without risking the pattern moving. I spent between 3 to 5 minutes ironing the pattern on my PCB's.

The larger the board the longer it will take and you must ensure you overlap the points where you apply pressure so no areas are left untouched. Once you are happy you have completed this part of the job, the board should be dropped into the bowl of hot water.

Peeling

If you wear thumb rings, take them off now or you may scratch off the pattern in this stage. After 10 minutes, lift the PCB out of the water and carefully rub the paper with your thumbs. It should start to roll off in thick strips. Remove all the thick layer first then place back in the water. You should remove the remaining paper layer by layer returning the board to soak in the bowl periodically. At some point you will break through to the copper areas. Clean off all the paper leaving behind the toner pattern only. You may notice that bits of track, pads, or nicks appear where the toner pattern hasn't stuck. Do not panic this can be repaired once all the paper has been removed.

If things have gone seriously wrong you will have to use the steel wool to clean off the pattern and start again. I have found that a hard toothbrush can be used to help remove the last of the paper. It can also help running your thumb nail backwards over hard to remove areas. One of the areas I would suggest you pay particular attention to is the pads. Particularly the component holes. Use your thumb nail, backward strokes, to clear the paper from the holes. This will pay dividends after etching, when you start drilling the holes.

Reparations

OK, all the paper has been removed and you have a fairly good transfer. However, some track has come off, pads missing or incomplete, small nicks in the traces, toner looks a bit thin here and there. All is not lost. Experience and practice will help minimise these problems and an etch resist pen will fix all of it. An etch resist pen resists the action of the etchant and as it's a pen, it is very easy to use to fix broken traces, pads and thin toner areas. You can draw lines with it and circles for broken pads or use it in a dotting action to rebuild the wayward areas.

Once you are happy you have fixed everything, you should check that all the paper has been removed from the pattern especially in areas where pads or lines are close together. If it is not removed you may end up with a 'short' where the paper has acted as etch resist shielding copper that should have been etched away.

Etching

Please observe all manufacturers safety notifications and data sheets, when using etchant.

Here's the fun part. My set up is as follows: I have a plastic ABS seed tray, that I use as my etching tray. It measures, 10″x 6″x 2.5″. This tray is sat in another, larger tray which at the appropriate moment is filled with hot water. I also have a bucket of water on hand and some plastic covered wire. First thing to do is pour the around 500ml of etchant into the small etching tray. Then pour hot water into the larger tray but not so much that the etching tray floats.

Prepare the PCB for etching: If the board is still oversize you should trim it to size with the hacksaw as before, being careful not to scratch or chip the pattern. Drill at least two holes in the PCB, close to one end. If there are mounting holes on your design then drill these. Now you need a length of plastic covered wire. This should be long enough so that when each end is fitted to the holes in the PCB the loop is large enough not to disappear under the etchant when the PCB is submersed. Place the PCB in the etchant. Whilst preparing the PCB the etchant has been warming up from the hot water in the outer tray. This will speed up the process considerably but is proportional to the area of the PCB you are etching. My 121mm x 67mm PCB took between 10 and 15 minutes to etch. At the start it will seem as if nothing is happening, for a while, so be patient. Gently agitate the board by lifting it up and down in the fluid. This will help clear off the copper 'sludge' as it degrades. If left, the sludge will slow down the process.

At some point you will notice definite thinning of the copper at the edges of the PCB and gradually whole areas of copper will disappear before your eyes. Don't be afraid to lift the PCB up out of the fluid to inspect it, particularly when the process is nearly complete. The instant all of the copper has been removed you should place the PCB in the bucket of water and swish it around to remove any residual etchant.

Well done! You have now successfully etched your first PCB. Allow yourself a moment to feel smug. Before we go on to the next stage, check that there is no bare copper left anywhere on the board, then tidy up and put the etching fluid in a safe place (not in a metal container), out of reach of children.

Drilling

With the PCB etched, it's time to perform the last major job. Drilling all the holes. This should be done in a systematic fashion to avoid changing drill sizes back an forth. Drill the larger holes first and work across the PCB in sections. An example is my PCB. 3mm mounting holes first, 1.5mm holes for the power wires, 1mm holes for larger component legs and hookup wires and finally 0.51mm for the rest. As you complete each group check the board before changing drill sizes, in case you have missed any holes.

The method of drilling is also important, particularly where components of a fixed pitch are concerned. You should eye up the drill to hole and then carefully touch the PCB surface with the tip of the bit. Let the board 'float' slightly and as the bit engages in the etched hole of the pad you are drilling, it will centre itself and, if not quite lined up the PCB, will move slightly to centre itself. Now hold the PCB firmly and drill through. Let the bit drill the hole without forcing it through. Move on to the next, eye up, let the bit move the PCB as it centres in the hole, hold and drill through. If any holes didn't etch, you will need to gently and accurately centre punch them. This, again, applies mainly to the fixed pitch component holes. Any holes that aren't reliant on fixed pitch can just be drilled freely. You will notice as you touch the PCB with the bit, that the tip will wobble around on the face. Let the tip settle then carefully drill through without excessive force. When you have drilled every hole, it is time to inspect the board for any holes you may have missed.

Finally

With all the hard work done, it is time to clean off the transfer material. You can either use the steel wool to work back to the copper; alternatively, acetone will clean off the toner. Once all the material is removed, clean the board in warm soapy water, dry, and avoid touching the copper traces too much. That's it. You now have in your hand your very first self made PCB. It is a thing of beauty and you can feel justifiably pleased with yourself.

One final note. You may find the the holes on the component side haven't cleared completely, particularly the 0.51mm holes. If so, spend some time with the correct sized bit in a pin vice and drill through from the component side to clear the 'swarf'. You can also dry fit some components to check that their legs fit the holes in the PCB.

For the beginner, all the above can seem daunting. Each stage is simpler than the overall guide and probably much easier than it reads. As mentioned earlier, for further tuition check out the tutorials available on YouTube (see Forum for links). There are slight differences in approach to PCB making, but all methods are valid.

Post Etching Tools

- **Dremal drill with stand or similar.**
- **A selection of HSS drill bits** and/or, tungsten carbide PCB bits. Sizes for components will range from 0.51mm, 0.9-1mm. For wiring you will need 1mm, 1.5mm.
- **A centre punch** (To be used carefully). This is to mark the component holes in the pads if the etching hasn't defined them. Holes that have etched will naturally 'guide' the drill bits.
- **Solder.**

Solder is of course an absolute requirement if we are to join everything together on the PCB. Currently the nature of solder is undergoing a change due to environmental considerations. Lead free solder is now fast becoming the norm. Unfortunately lead free solder is not as easy to use as the traditional lead/tin mix. Practice is essential to get to know the methodology of it's use. A small pot of 'tinning paste' should be purchased if you have to use lead free. This is used to 'tin' the tip of the iron prior to making joints on the PCB. It is suggested that you hunt around for lead tin solder if possible. Rapid Electronics still stock 500g reels of lead/tin solder, so stock up while you can. Lead free solder can be purchased from Maplins and possibly your local hardware store, in small plastic tubes. One should suffice but buy two -just to be sure. You should buy either type with an Swg (thickness) of 22Swg – 24Swg and it must have a 'flux' core.

It's built but doesn't work. What do I do?

First up do not panic. The Forum is the place to ask questions if you have problems, but lets cover some of the basics First things to check are: Battery charged up and in good condition? Mic and speakers plugged into the correct sockets? (It has happened). Power plugged in to the correct socket? Lights plugged into the correct socket? Power supply polarity iorrect? Remember the plugs and sockets are different sizes to help avoid this, so you may have put the plugs on the wrong item.

If all the above are OK then you need to look at the PCB...

1. Check the IC's are all inserted in the right sockets and orientated correctly.
2. Check all electrolytic capacitors are orientated correctly.
3. Check the wiring of the mic and speaker cable at the PCB and sockets. The mic wire has to be fitted correctly as the mic has a 'polarity'.
4. Check the wiring to the variable resistors.
5. Check the power wires are orientated correctly at the PCB and at the sockets. Also check the power plug for polarity.
6. On the track side of the PCB check for dry joints, (joints that have not been soldered).

Also check for short circuits where solder may be bridging separate joints or traces.
Any of the above can happen and has happened to me.

Set-up and use of the unit

Your voice mod, when fitted, will no doubt be surrounded by other electrical devices and wiring therein. What is important here is that power and audio wires do not run together (side by side) as this can cause the pick up of spurious emissions. They can cross at 90 degrees though so do bare this in mind.

Now we need to look at 'tuning'

Before powering up the unit ensure that the EQ trimpots are all fully CCW then turn them CW by around 1 degree. This just ensures the wiper hasn't run off the resistive track. Also ensure VR7 is fully CCW. The set screw on VR7 can be turned 25 full rotations. The wiper is on a slip clutch so when you reach the end of travel in either direction you can hear a very faint click. So when you turn this CCW listen for the click and VR7 will then be at it's start point. Now power the unit up. You should hear a slight buzzing at the speaker. Speaking into the mic will be to no avail. Now with a small flat tip screwdriver slowly turn the set screw of VR7 CW whilst talking. Have the speaker around 6" - 8" in front of you, volume turned up to maximum, with the back of the speaker facing you. It does go with out saying if the speaker is not cased your going to get feedback very early on probably leading to low volume. So seal the back of the speaker either with acoustic lagging, build a box or both. Now keep talking whilst turning the set screw and you will gradually hear your voice rising in volume. Keep going until you start getting high pitch whistling. This is feedback and can be reduced by turning the set screw CCW until the feedback stops. At this point your pretty much sorted.

The use of the tone controls

Bass is the left dial, nearest the loom. Turning it clockwise will increase the low frequency's present in the output. Mid range is the middle dial. Turning it clockwise will increase the mid range frequency's present in the output. Treble is the right hand dial. Turning it clockwise will increase the high frequency's present in the output. These should be used as set and forget items as the cermets have a finite lifespan. Once your happy with the sound of your voice mod leave them be. Note that depending on Speaker, Mic set-up that the treble may not have much, if any, usable range due to it being the high frequency's that cause most feedback. Mid range will have a greater amount of range, but, again you will reach a point where feedback occurs. The bass trimmer will have the greatest usable range. Ideally the unit should be positioned so that you have easy access to it and it's controls and can see the LED's so you have visual references to power being present and what the dome lights are doing.

This PDF MKIIIb voice mod has seen some great improvements to it's design since the original version. One of these is the improved suppression of feedback which means we have more options for positioning the speakers. With the NSD the main options are in the fender (front or rear) and in the neck bin. Original series Daleks could have the speakers positioned either in the fender area similar to the NSD in the neck bin or in the shoulders with holes cut in the shoulder wall behind the belts so they cannot be seen. The suggested speakers are 4 ohm or 8 ohm no smaller than 100mm and no larger than 125mm in diameter.

You can have either one speaker or two speakers in parallel. See the diagrams below for the differences in the Watts you can get. Any speaker that is open backed must be sealed/covered with material that will suppress the propagation of sound into the Dalek which may cause feedback. This can be carpet, hard foam rubber, Plastic bowls as used for microwave Christmas puddings which should have any voids filled with any of the above and even cover the exterior with the same.

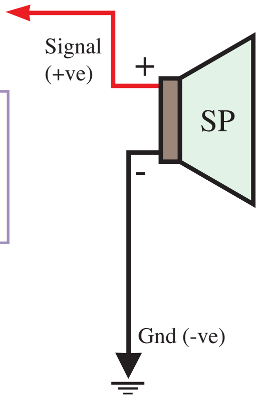
As you can see in the illustration (top of following page), the output in Watts is effected by the speaker rating. So bare this in mind when you choose your speakers. They must be capable of handling the expected Watts. Note that with speakers in parallel the speakers must be identical. There is also provision on the PCB for fitting a 'line out' facility. This bypasses the on board amplifier and the audio can be routed to a car amplifier for even greater output. If you use this option unplug any speakers from the unit itself but leave the on board volume level alone as it is what drives the sound to light of the unit. wiring up the speakers. This is to do with synchronising the speaker coil to the signal.

Single speaker:

2 Ohm @ 12V = 10.5W

4 Ohm @ 12V = 8W

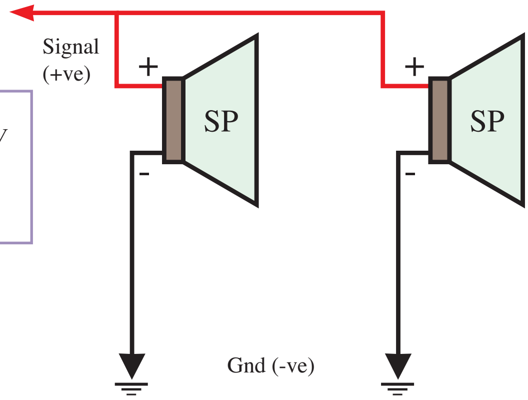
8 Ohm @ 12V = 5W



Parallel speakers:

4 Ohm @ 12V = 10.5W

8 Ohm @ 12V = 8W



A modification to the voice mod for owners of Movie Daleks.

Movie Daleks have a bit of echo/reverb to their voices. However the PDF voice mod only deals with the modulation of the voice. Fear not! There is a solution, should you wish to have the authentic voice for this type of Dalek. A second hand electronic music effects module can be used to add the required effect. Names that spring to mind, (as I have both), are the Zoom 2001 and the Zoom 606, Others are available. These are effects modules for guitars, usually, and one of many effects they have are echo/reverb.

Method 1: (uses external amplifier)

If you have fitted the line out facility you are half way there already. If you haven't then you may consider this option and the following option. Plug the line out of the voice mod into the audio in of the module then from the audio out of the module into an external amplifier such as a small in-car amplifier. Turn every thing on and set the reverb to taste.

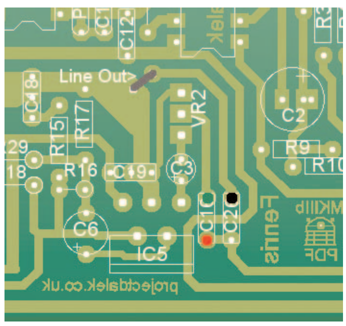
Method 2: (uses on-board amplifier)

All that is involved on the PCB is a slight rewire. Basically the rewire will separate the on board amplifier from the modulated signal. These two halves will have their own audio socket. The socket on the audio side should be referred to as 'Signal Out', the signal being your modulated voice. The other socket should be referred to as 'Signal In'.

The wiring on the PCB takes place at C16, C26 and the 'line out' header and is very simple.

If C16 and or C26 have been fitted then remove these. The two holes for header point, for the line out should be bridged with a piece of wire. Next you need two lengths of single core audio cable. The length of these will be determined by where you intend to fit the two new sockets. Now strip the 20mm of outer sheathing from one end of each. Twist the shield wires of each. Now strip 5mm of sheathing from the inner core wire, again, twist each end. Tin these ends. Now back to the shield wire of each cable. Untwist each and separate each group in half. Now twist one half of each group and then cut it off ensuring there are no stray whiskers. Now the remaining shield wires of both cables should be brought together and twisted into a unified whole. Lightly tin the resultant bundle.

Now it is time to fit the cable to the PCB. There are 3 points of contact and to aid you these have been colour coded. The joint shield wire is fitted in the Black coloured Hole. Do use a small piece of the outer sheathing you removed earlier to cover the bare shield wire to prevent 'accidents'. One core wire is fitted in the red hole and the remaining core wire is fitted into the green coloured hole.



The cable fitted to the red hole is the signal out. The cable fitted in the green hole is the signal in. All that remains to do is fit the sockets of your choice to the other ends of the cables, bearing in mind that the centre terminal is the one the inner core of the cables is attached to and the outer terminal is for the shield wires. What you need to do now is make some patch cables to use to link the voice mod to the effects module. You can also make a patch cable that will bridge the two sockets for normal use.

The only advantage of the second method is that it negates the use for an additional, external amp.

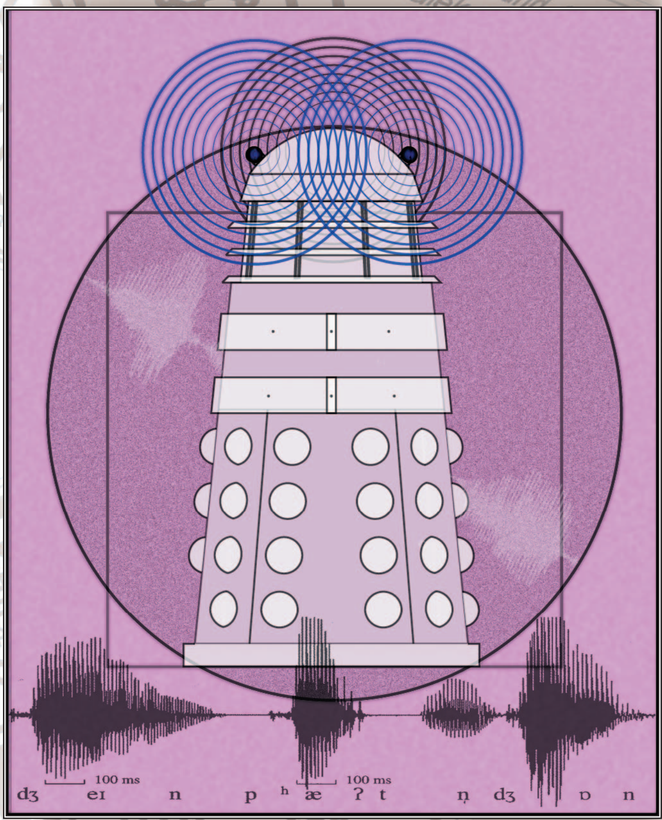
Talking like a Dalek

You do have to act the part. The voice modulator only adds an effect to your voice. So if you have a broad Yorkshire accent, for instance and talk normally into the unit you will get a Yorkshire accent with a funny effect. Anger, arrogance and downright hostility are the hallmarks of the Daleks. Remember they seek to destroy all other non Dalek life forms, in their quest for supremacy of the universe and have nothing but contempt for other species. So anger, with a bit of arrogance thrown in and a side order of haughty superiority will see you on your way. Add in the 'mechanical' structure of their speech pattern and you should be terrorising small children with no problems at all.

Parts List

Part	Value	Type	Pitch	Qty	ID#				Vendor
C1	33uF	Elec	3.5mm	1	11-3088	11-3760	11-3744		Rapid
C2, 4	10uF	Elec	2.5mm	2	11-1554				Rapid
C3, 5	4.7uF	Elec	2mm	2	11-1522	11-2929	11-1290	11-1522	Rapid
C6	22uF	Elec	5mm	1	11-1514				Rapid
C7	470uF	Elec	3.5mm	1	11-2997				Rapid
C8, 10	2.2uF	Elec	2mm	2	11-1553				Rapid
C21	220uF	Elec	5mm	1	11-1175				Rapid
C9, 12, 17, 18, 19, 20, 22, 24	0.1uF(100n)	Poly	5mm	8	10-3260				Rapid
C11, P1	1uF	Poly	5mm	2	10-5836	11-0656	11-0688		Rapid
C13	47nF	Poly	5mm	1	10-3258				Rapid
C14, 23	4.7nF	Poly	5mm	2	10-3248				Rapid
C15, 16	22000pF(22n)	Poly	5mm	2	10-3254	10-2242	08-0916		Rapid
C25	1uF	Ceramic	5mm	1	08-1100	11-0656	11-0688		Rapid
R1, 21, 22, 24	1k		0.25	4	62-0370				Rapid
R2, 23, 25	10k		0.25W	3	62-0394				Rapid
R3, 4	220k		0.25W	2	62-0426				Rapid
R8, 9	470k		0.25W	2	62-0434				Rapid
R6	33k		0.25W	1	62-0406				Rapid
R11, 32	12k		0.25W	2	62-0396				Rapid
R12, 33	3k9		0.25W	2	62-0384				Rapid
R13, 34	1k8		0.25W	2	62-0376				Rapid
R14	8k2		0.25W	1	62-0392				Rapid
R15, 18	1R		0.25W	2	62-0303				Rapid
R16	220R		0.25W	1	62-0354				Rapid
R17, 29	10R		0.25W	2	62-0326				Rapid
R7, 10, 20, 31	100k		0.25W	4	62-0418				Rapid
R19, 26, 27	100R		0.25W	3	62-0346				Rapid
R28	2K2		0.25W	1	62-0378				Rapid
R30	470R		0.25W	1	62-0362				Rapid
R35	82k		0.25W	1	62-0416				Rapid
VR1, 3	10K Lin		24mm	2	65-0515				Rapid
VR2	10k Log		24mm	1	65-0615				Rapid
VR4, 5	100k		cermet	2	67-0446				Rapid
VR6	500k		cermet	1	67-0450				Rapid
VR7	500k		bourns	1	68-0432				Rapid
IC1	LM358			1	82-0258				Rapid
IC2	M74HC4066			1	83-0392				Rapid
IC3	ICM7555IPA			1	82-0774				Rapid
IC4	4N35			1	58-0816				Rapid
IC5, IC6	TDA2003			2	TDA2003				Cricklewood
D1, 3	1N4148			2	47-3416				Rapid
D2	P1000A or BY80-200			1	47-2964	47-3832			Rapid
D4	5mm Red LED/Limiting Resistor			1	56-1550				Rapid
Q1, 2	BC548B			2	81-0466				Rapid
Q3	BC558B			1	81-0184				Rapid
Q4, 5	IRF540			2	47-0317				Rapid
PCB				1					Fenris
14 pin Dip Skt				1	22-1721				Rapid
8 pin Dip Skt				2	22-1720				Rapid
6 PIN Dip Skt	Available only in tubes of 80off			1	22-0118				Rapid
Heat sink	12.7x30x25			2	36-0235				Rapid
Thermal pad and bush		PK10		2	38-0245				Rapid
Control knobs				3	32-0295				Rapid
SPST toggle switch				1	75-0082	75-0083			Rapid
Power skt	2.1mm			1	20-0878				Rapid
Light skt	2.5mm			1	20-0880				Rapid
Spk Skt Stereo	3.5mm			2	20-0135				Rapid
Pwr plg	2.1mm			1	20-0882				Rapid
Light plg	2.5mm			1	20-0888				Rapid
BOX	150x80x50mm			1	30-1839				Rapid
1 core shielded shielded Mic Cable	250mm			1	XR12N				Maplins
Twin Power Cable 6A	300mm			1	XS70M				Maplins
Hookup wire: Red	100mm			1	FA31J				Maplins
Hookup wire: White	100mm			1	FA29G				Maplins
Servo cable: Tri-colour	300mm			1					Model shop

Please Note: Capacitor P1 is only required if you want the 2 extra frequency's. See preface for details. Further to this point, if you go for the tantalum bead option these have a tolerance of 20%. This means they vary either side of the actual value. i.e. the minimum value could be as low as 0.800nF and the highest 1.2uF, which will affect the frequency range.


100 ms
100 ms
dʒ eɪ n pʰ æ ʔ t ŋ̍ dʒ ɒ n