General Construction Notes

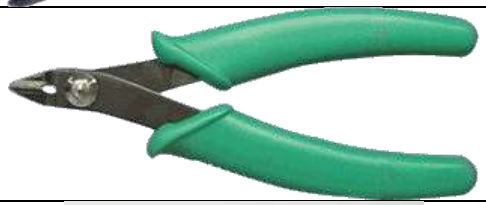
This guide is intended to be a useful source of information for all builders of our kits.

Please note that although we have tried to make sure that no errors have occurred in the preparation of this document, we do not accept any liability for any inconvenience or losses of any kind caused by the information presented in this document. If you do find any errors or obsolete information in this or any other of our documentation please let us know by email to elby-designs@bigpond.com.
.

General Construction Notes

General Tools

The following basic tools are recommended for constructing most electronic kits:-

Small, flat-bladed screwdriver		A 3mm flat bladed screwdriver will be fine for most small-scale work.
Small, cross-head screwdriver		A Number 0 Phillips screwdriver is recommended. Note that Phillips and Pozidrive are different and you should use the relevant screwdriver for each screw type.
Small side-cutters		You can get side cutters with either a flat cut or a recessed cut as shown here. 
Small snipe-nose pliers		
Small pliers with serrated teeth grips		

General Construction Notes

Temperature-controlled Soldering Iron		The iron should be, typically, between10 and 30 watts. There are a wide variety of tip shapes but a small chisel style (around 1.2mm tip) should suit most applications.
Solder-sucker		Hopefully you don't need to rework a board but if you do then a solder sucker is useful for clearing out pad holes after removal of a component. Solder wick is also a lower-cost option.

In addition, for testing and calibration you will need one or more of these items:

1. Multimeter: should be capable of reading voltage, current (typically no more than 500mA) and resistance. A continuity or diode test function is also useful.
2. Oscilloscope: a general -purpose model is fine for most work required here
3. Frequency-counter: for most work here the oscilloscope in (2) is usually adequate for measuring frequencies
4. Crimp Tool: some projects use MTA terminals for external wiring. Crimping for these connectors should, ideally, be made using a proper crimp tool

General Construction Notes

Bills Of Materials (BOMs)

The Bill Of Materials (BOM) is the master construction file and is, in theory, the only document required to build a product. The only additional information that is sometimes needed includes notes on specific assembly points, modification details and calibration procedures which are all usually included in the Build Guide for the product.

A typical BOM can be seen here with an extract shown to the right.

The BOM lists all items by component type (Capacitor, Diode, Resistor etc) and by numerical value within that category.

Each line, which has only one component type/value, there is a QTY field indicating how many of this part go on to the specific PCB being built, followed by a list of DESIGNATORS which refer to a specific point on the PCB. A PACKAGE field describes the physical type and size of the component.

Bill of Materials for ES20 - 1973 VCO [4mm] Full Kit V0.0 - Main Board

Item	Description	Qty	Designator	Package
	CAPACITOR			
1	33pF 5% Ceramic	1	C104	RAD-0.2N
2	47pF 5% Ceramic	1	C102	RAD-0.2N
3	82pF 5% Ceramic	1	C202	RAD-0.2N
4	1nF 63V Polyester	1	C103	RAD-0.2
5	10nF 63V Polyester	1	C101	RAD-0.2
6	10nF 5% Ceramic, Medium K	1	C305	RAD-0.2N
7	100nF Ceramic	2	C303, C304	RAD-0.2N
8	4.7uF 50V Electrolytic	1	C201	RB-D5.0-P2.0
9	10uF 35V Electrolytic	2	C301, C302	RB-D5.0-P2.0
10	47uF Electrolytic	2	C106, C203	RB-D6.3-P2.5
	CONNECTOR			
11	IDC16 Receptacle	2	J101, J102	-
12	IDC16 Box Header	1	J301	2.54MM
13	14-pin IC Socket 0.3inch	1	U101	DIP-14
	DIODE			
14	1N4148	3	D201, D202, D203	DIODE-0.3
	LINEAR			
15	LM3900 - Quad Norton Amplifier	1	U101	DIP-14

In products comprising multiple PCBs (sometimes referred to as sub-assemblies), the BOM will list the parts for each PCB separately with the heading at the top of each page indicating which PCB is being worked on. In the above example which is a 3-PCB module, we are working on the MAIN BOARD

General Construction Notes

Printed Circuit Boards (PCB)

All our PCBs are high-quality double-sided boards and use plated-thru holes.

Being double-sided means that the PCB has tracks on both the TOP and BOTTOM layers of the PCB. Where required, these tracks 'join' each other using a VIA which comprises 2 pads in corresponding positions on both the TOP and BOTTOM layer which are electrically connected by a hole through the board and which is then made electrically conductive by electroplating. These VIA's do NOT require the builder to apply solder to them.

The PCB's use a RoHS compliant lead-free HASL surface finish. This has a good shelf-life and provides a good wetting surface for soldering. Although it is quite possible to use lead solder, better results are achieved using lead-free solder. HASL boards are not suited for fine-pitch surface-mount devices.

A silk-screen is used on all boards to identify the location of each component. Being double-sided also means that it is possible to place components on both sides of the board. When components are to be mounted on the BOTTOM layer, a silk-screen will ASLO be printed on the BOTTOM layer i.e. ALWAYS mount the components on the side that their silkscreen is.

Pots and Rotary Switches

Our kits mainly use ALPHA 16mm pots which come with an 8mm 18-tooth splined shaft and have an optimum length shaft that does not require shortening. We occasionally use ALPHA 9mm pots which have the same shaft.

The pots come with a locating or anti-rotation pin which should be removed by snapping it off with stout nosed pliers or cutting it off using a heavy-duty cutter (please do not use your normal cutters for this job :).

General Construction Notes

Component preparation.

Most components in the kits we supply are designed to be mounted directly on to the board requiring only minimal pre-forming in some cases (mainly resistors and diodes). If, however, a component is supplied or used that does not match the footprint on the pcb then you will, of course, need to form the legs so that the component can be fitted. A pair of small snipe-nose pliers should, ideally, be used for this task. The leg to be formed should be held by the pliers at a point immediately before the position the bend is to be made (i.e. the pliers will be holding the leg on the side nearest to the component body). Using your fingers or another pair of pliers or suitable tool, bend the leg to the desired angle. For axial resistors this would typically be 90°.

Sometimes a component such as a tantalum capacitor may be supplied, for example, with 0.1" pitched legs but needs to be fitted in to a 0.2" footprint. In these cases you should form both legs as described above. As always, pay careful attention to minimize the stress on the junction between the leg and the body of the component.

Light Emitting Diodes (LED):

Some modules require that the LED be mounted at a greater distance from the pcb than is achievable by direct installation of the LED. In these instances our kits will include a length of red and black wire and some heatshrink sleeving. Prepare these LEDs as follows:-

1. Cut the ANODE (longer leg) to a length of about 7mm
2. Cut the CATHODE (shorter leg) to a length of about 5mm
3. Strip, twist and tin approximately 3mm of one end of each of the leads
4. Solder the red wire on to the ANODE leg and the black wire on to the CATHODE leg. Be sparing with the solder so that there are no large lumps
5. Cut the heatshrink sleeving in to 10mm lengths and slide one over each wire.
6. Use a heatgun or similar heating tool to lightly shrink the sleeving
7. Cut the longer lead so that both wires are the same length
8. Strip and twist about 5mm of the end of the leads
9. When ready for assembly on to the panel, insert the LED making sure it snaps in to place
10. Insert in to their respective pads (Red = ANODE, Black = CATHODE) and solder in to place

General Construction Notes

Component installation

It is recommended that you install components by height starting with the smallest components (wire links and diodes) first and gradually building up to the tallest components. Do not try to fit too many components at the same time otherwise you may have difficulty accessing the solder points on the components, this is more specifically true for components with long legs such as resistors and capacitors.

Although not essential to the operation of the unit, it is desirable to keep components flat to the board (unless the design specifically requires a component to be mounted `off the board'). For long legged components we suggest bending the legs at an angle after they have been inserted in the board. An angle of around 45° is fine as it still allows the component to be removed fairly easily should the need arise. An alternative method is to temporarily hold the components to the board using a low-tack tape. A few components are positioned on the board and then a strip of tape is laid over them, flip the board over and solder.

A solution for when you intend to build a number of boards is to get a piece of, about, ½" foam about the same size at the board you are going to work on. Glue a piece of stiff card on to one face of the foam. Rest your pcb on some support so that the long components legs can protrude through the pcb without hitting anything. When you have placed a number of components, lower the foam on to the pcb (foam side down). Grasp the pcb and foam at each end and flip them over placing it on a flat surface. Carefully tack one leg of each component and once done, flip the pcb over and check that all components are sitting flush to the board. Flip the board back and solder all the remaining joints making sure you reflow the `tacked joint' last to ensure that it has soldered properly. It is crucial with this method that you adhere to the lowest to highest rule here otherwise you will find components end up standing off the board.

If you intend to do large amount of pcb assembly then it is worth investing in a pcb assembly jig such as that shown here.

This works on the same principle as the foam option above but uses a clamp mechanism to hold the foam firmly against the board.

Earth\Ground Tag

Many Panther kits include an Earth Tag which is used to provide a ground connection to the front panel to help eliminate hum and other possible noise problems that might occur with a floating panel (just because the panel is mounted to metal rails and may be using metal screws for fixing does NOT mean you have an electrical connection to the metalwork!!!!).

To mount the tag you first need a short piece of wire. Use one of the legs cut off a resistor (approximately 20mm will be fine) and form a small hook at one end. Insert this hook in to the small hole on the tag, clamp tight and solder.

Slide the larger hole over the shaft of a pcb mounted pot (the Panther pcbs have a hole marked PANEL-GND next to a suitable component) and insert the free end of the wire in to the PANEL-GND hole. The tag should be sitting with the wire coming away from the pot at about 70-90 degrees and parallel to the panel.

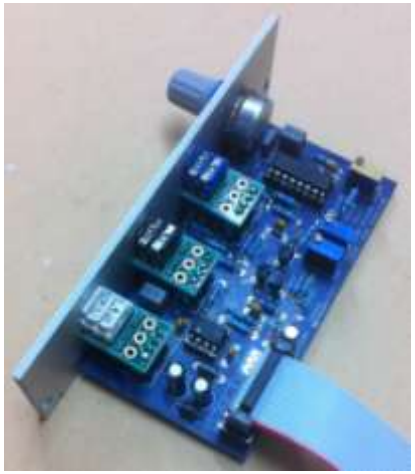
When the assembly is mounted to the front panel, the tag will form an electrical connection to the rear of the panel.

Not all modules will have a pot which is the ideal mounting component so sometimes you will need to fit the EARTH-TAG over a jack. It is important to ensure that when soldering the EARTH-TAG in to place that the EARTH-TAG is sitting flush against the body of the jack and not caught over the thread.

If the module contains a toggle switch that is directly mounted to a PCB then an EARTH-TAG may not be supplied as the metal body of the switch is used for the grounding function.

Panther Build Options

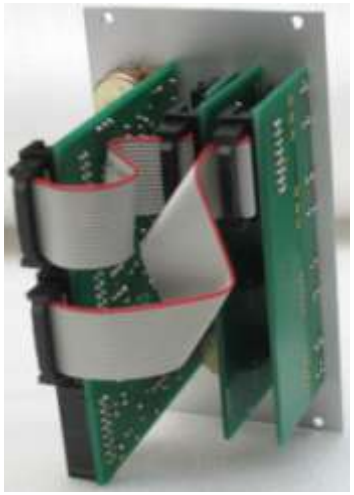
Panther modules adopt one of 2 main build options:-



1) Cable-Board.

In this method a multi-way IDC cable is plugged in to an IDC connector on the Panther Support board and the other end is plugged in to mating connector, usually on the main board.

The picture at right shows the ES79 Ring Modulator module. The large blue board is the 'main board'.



It has a pcb-mounted pot at the top and then a number of Panther Support boards.

The BOM will indicate which cable length is to be used for each connection. The mating board will have its connectors labeled identifying which Panther Support board it should be connected to.

General Construction Notes

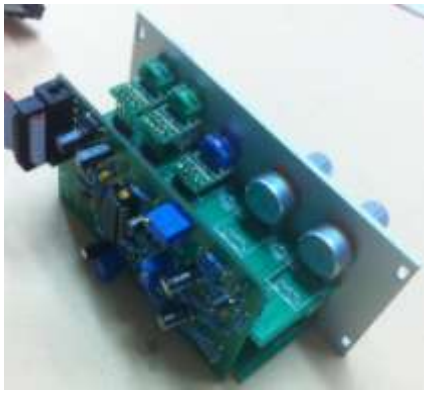
2) Board-Board

With this method all the panel components are mounted using Panther Support boards either directly and/or using Panther Carrier boards.

The 'main board' is then 'plugged' in to the back of the boards as shown to the right in the ES20 1973 VCO..

IDC Receptacles (pcb mounted IDC sockets) are mounted on to the back of the 'main board' and mate with IDC connectors on each of the Panther Support boards.

You can see the IDC Receptacles at the bottom of the 'main board' in the photo to the left.



Modules built using this method are all 59mm with only a few exceptions (the IF112 Double Deka is only 40mm deep).

All early Panther modules were built using the 'panel mount' method but many are being rebuilt using the 'backboard mount' method.

None of our Panther boards include mounting points making them harder to use in DIY builds that do not utilise our Panther Support boards or, where applicable, the onboard pcb-mounted panel components.

Board-Board Interconnect. In this method the main board is mounted parallel to the back of the Front Panel and has IDC Receptacle sockets that mate with each of the IDC connectors on the Panther Support boards. As well as eliminating all but one of the IDC ribbon cables (you still need one for connecting to the power supply), this method also sets the overall module depth to, typically, 60mm

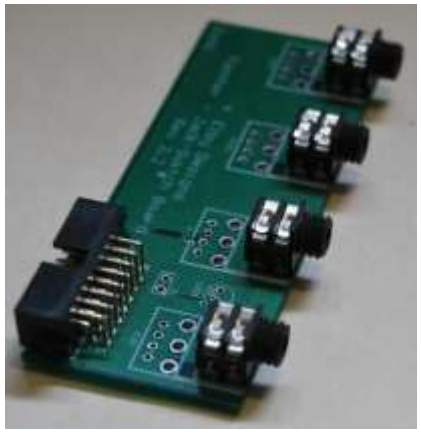
Panther Support Boards

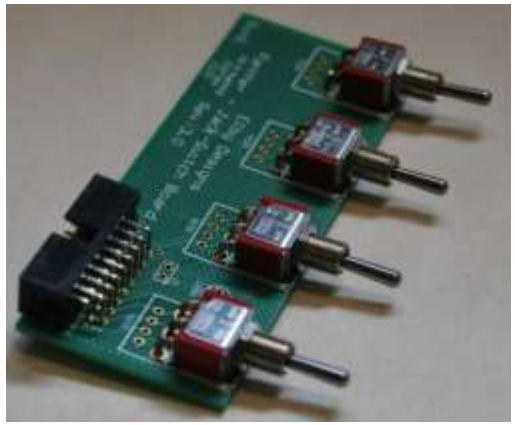
To help minimize the amount of wiring that is often needed to connect panel mounted components to the circuit board the Panther family make extensive use of Panther Support Boards. There are, currently, 5 members of this family:-

1. Panther Jack-Switch Board
2. Panther Jack-LED
3. Panther Pot Board
4. Panther Support-5
5. Panther Carrier Board

Panther Jack-Switch and Jack-LED Board

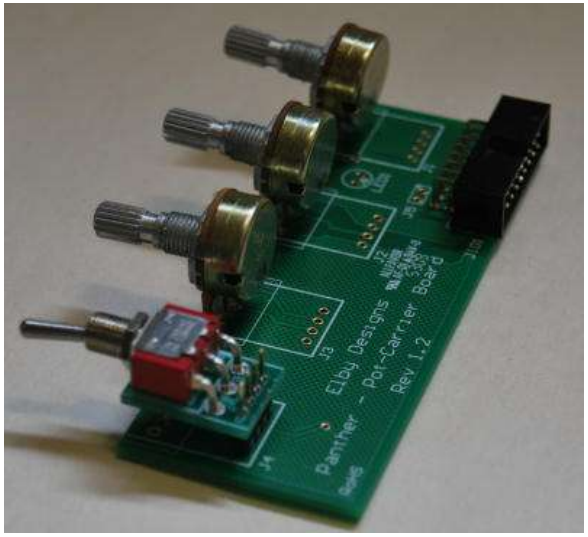
These boards can support any mix of up to 4 jacks and switches. The 2 pictures below show the boards dressed with 4 jacks and the other with 4 switches





While the Jack-Switch board will accept a single LED in one of 3 positions, the Jack-LED will accept up to 4 LEDs and has 7 different positions for them (2 different positions for the 'top' 3 LEDs and one position for the 'bottom' LED..

Panther Pot Board

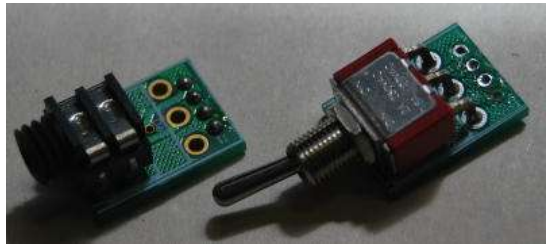


This board supports pots and, optionally, a mix of jacks and switches. When adding jacks or switches to this board they will (nearly) always be done by mounting them using the Panther Carrier Board. The picture shows this board with 3 pots and a switch.

Panther Support-5

This Support board provides for a an alternate panel grid layout that now provides for 5 components vertically in a column. It is a multi-component board accepting a variety of combinations of pots, jacks and switches.

Panther Carrier Board



To allow components on the front panel to be aligned vertically within a column, it is necessary to raise the lower height components such as jacks and switches. This is achieved by mounting the component on to a carrier board which is then held at the correct height using an extended header.

Two header heights are used in the Panther modules. A 20mm header is used for raising a jack or switch in to alignment with a pot and this can be seen in an earlier picture under the Panther Pot Board. The 20mm is also used where a design requires, for example, 2 jacks to be mounted on the same row and symmetrically offset from the pot centre line, The picture to the left is of the CGS738 Mangler and shows the header being used to allow for 2 jacks to be mounted in the same row.

IDC Receptacle Sockets - Installing

Correct alignment of IDC connector

When mating the back board of a module on to the front panel support boards, it is important to check for the correct alignment of the IDC Receptacle in to its mating IDC Headers.

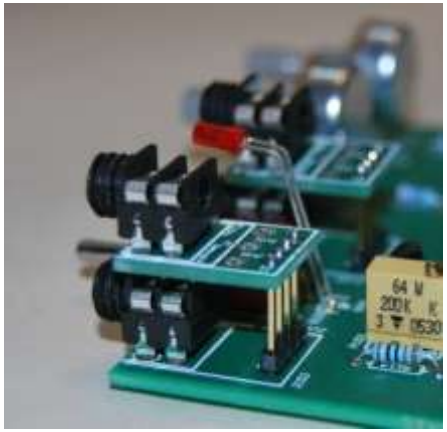
In the photo to the right the connector are misaligned and you can clearly see the pins of the header through the inspection window of the header.

On the left, the alignment is correct and the header pins are not visible.

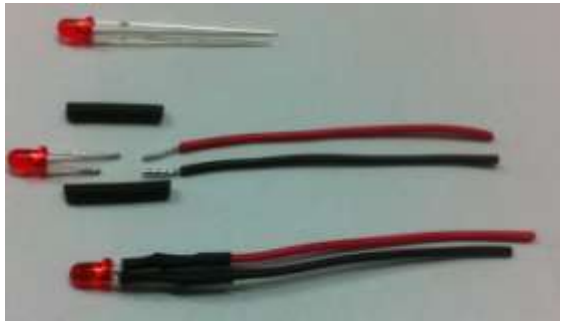
It is important to check both ends of the connectors.

Incorrect alignment of IDC connector

Mounting LEDs



Where possible, PCB designs allow for an LED to be directly mounted to the PCB. We recommend pre-forming the legs to reduce stress on the LED body and to provide some flexibility in the mounting after the LED has been soldered in to place



In some PCB designs the LED has to be positioned further back from the panel and it may not be possible to fit the LED due to the legs not being long enough. In these cases we supply 2 pieces of wire and heatshrink to create 'extended legs' as per the photo at right. See earlier notes regarding assembly.

General Construction Notes

Mounting Switches

When mounting switches on to any of the Panther boards you should fit one nut to the switch before assembly. Finger tighten the nut and then, if required, back off a fraction so that it has a flat face parallel to the pcb.

The body of the switch should then be adjusted to be parallel to the pcb. When mounting single-pole switches directly to any of our boards other than a Carrier board, you should endeavour to finish with the component legs flush with the solder side of the board.

3D Model

Mounting Transistors, Diodes, capacitors and ICs

For TO-92 transistors match the flat side of the device with that shown on the PCB legend. Push the transistor into place but don't push too far. Leave about 0.2" (5mm) of the leads visible underneath the body of transistor. Turn the board over and slightly cinch the two outer leads on the flip side, you can leave the middle one alone. Now solder the middle pin first, then the other two pins.

The TO-92 footprint has an extra 'centre pad' which can be used for transistors that have a staggered (triangular) formation of the legs. You can also use this pad instead of the 'centre inline' pad for inline formed legs if preferred.

Diodes can be treated much like the resistors. However, they must go in the right way. The cathode is marked with a band on the body of the device. This must align with the vertical band on the board.

Polyester capacitors are like little blue, yellow or red boxes. Push the part into place up to the board's surface. Little lugs are sometimes incorporated in to the body to leave enough of an air gap for any flux wash to work. Cinch and solder the leads as you would resistors.

The 0.2" pitch ceramic plates need to be treated with a little respect. Don't bend them to much once you have soldered them in. Do trim down the leads with wire cutters, even if they don't have that much to chop off.

General Construction Notes

The smaller electrolytic capacitors can be mounted up to around 0.2” above the board, with the legs slightly formed. Don’t force the capacitor to sit hard against the board. Electrolytic capacitors are polarised, and may explode if put in the wrong way. Oddly, the PCB legend marks the positive side with a ‘+’, although most capacitors have the ‘-’ marked with a stripe. Obviously, the side marked with a ‘-’ must go in the opposite hole to the one marked with the ‘+’ sign. Most capacitors usually have a long lead to depict the positive end as well.

Most ICs have a mark or indentation in the moulding indicating the end at which pin 1 is. Another general rule-of-thumb is that the labelling on the top of the IC is legible left-to-right when pin 1 is at the lower left corner

Once all the components have been mounted and soldered you should check that all the legs are neatly trimmed to a maximum of 2mm. Give the board a good clean with a pcb cleaner compatible with the solder flux that has been used. This is mainly to remove all excess flux which, if left, may eventually start to erode at the board and components.

Mounting the Pots

The first thing to do is to check your pot values. Different manufacturers use different coding methods to define the `law’ of the pot.

ALPHA pots use the following:-
A = Logarithmic
B = Linear
C= Anti-logarithmic

So, for example, a 100K Logarithmic pot from ALPHA will be marked 100KA whilst a 50K Linear pot will be 50KB.
Next, most pots have an anti-rotation tab which is intended to locate in to a mating hole on the front panel, see earlier notes about removing these.

Doing one pot at a time, fit each pot into the appropriate holes in the PCB. Solder the center pin of the pot leaving the other two pins. Now check if the pot is standing square to the pcb. If it is not, simply reheat the soldered pad to allow you to move the pot into the correct position. Don’t leave your iron in contact with the pad for too long as this will lift the pad and could also damage the pot by getting too hot.

When you are happy with the location, you can solder the other two pins of the pot. We recommend trimming the centre leg to about 1mm above the pcb to prevent possible contact with adjacent components when used on our Panther Support Boards.

Vias

These are the little solder pads with no legend that seem to attach to no component. They allow copper tracks to swap sides on the circuit board. There is no need to fill these with solder and in the case of most of ELBY Design boards this is not possible because they are tented (i.e. they have a coating of solder-mask over them) to prevent liquids such as solder getting in to the hole, as well as providing a solid surface over the holes for any silkscreen printing and ensure that the printing remain legible.

Interconnections

Interconnections between multiple boards and/or panel components can be done by a variety of ways:-

1) direct wiring, which simply means wires soldered onto the little pads on the board which then go to the socket, pot or switch individually.
2) 0.1” jumpers, which are flat one piece items that carry several signals to another board.
3) 0.1” KK or MTA flexible interconnections, which are header and plug based systems. It is only the headers, the little white plastic things with pins that stick upwards, that are normally fitted at the board populating stage.

Wherever possible our pcb designs retain a common 0.1”-pitched spacing on the terminals of the connectors allowing you select from the many options the one that best suits the needs or your budget. If using the direct wiring method it is important to ensure that once a joint if finished that the sleeving of the wire is still close to the solder joint and that you have carefully trimmed any excess from the solder side.

Vertical Jack-Carrier Boards

Start by preparing the 1x4 Headers. Slide the support bar on each header towards the bend in the header pins – in the picture to the right the right-hand header is after this operation has been completed.

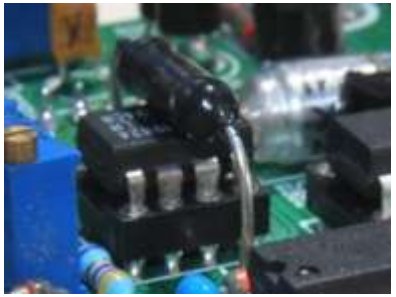
General Construction Notes

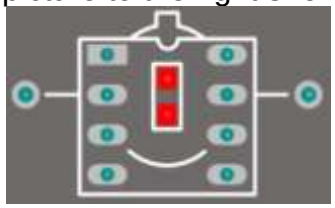
Mount the header on to the Carrier pcb ensuring that the header bar sits firmly and squarely against the pcb and the bar is parallel to the pcb as shown here.

Fit the jack socket.

Tempco

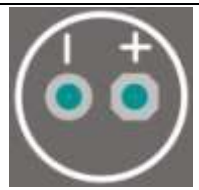
Tempcos usually need to be mounted in close thermal contact with a main component on a board. The picture to the right shows a PT146 style Tempco mounted across an IC using the pcb footprint below.



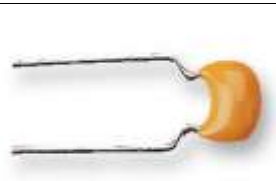
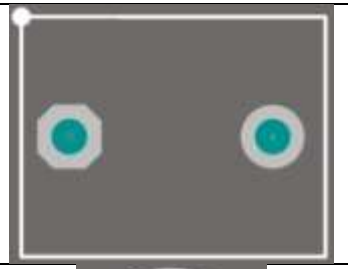
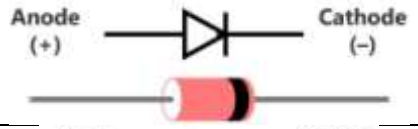


A small amount of heatsink compound can be applied between the body of the ic and the Tempco to improve thermal bonding

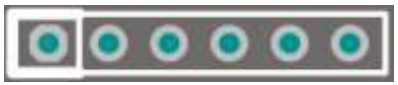
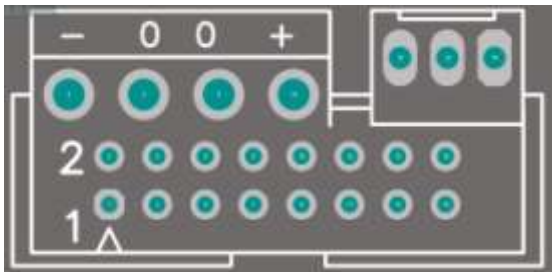
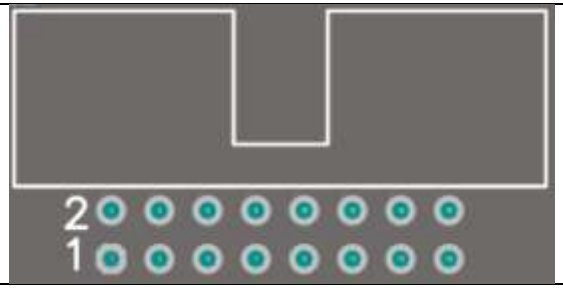
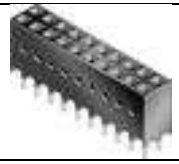
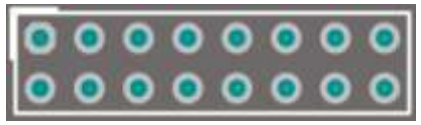
General Construction Notes

COMPONENT	IMAGE	TYPICAL PCB FOOTPRINT
Capacitor Ceramic		
Capacitor Electrolytic		
Capacitor Bi-Polar		
Capacitor Polyester		

General Construction Notes

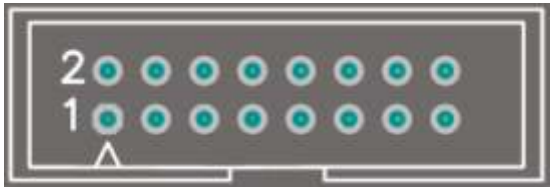
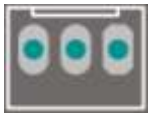
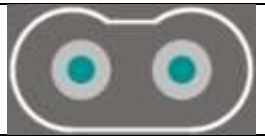
Capacitor Monolithic Ceramic		
Capacitor Polypropylene		
Capacitor Tantalum		
Capacitor Polystyrene		
Diode Signal - Zener		
Diode Power		

General Construction Notes

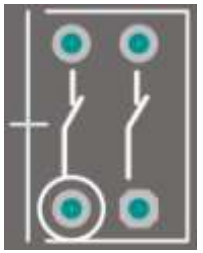
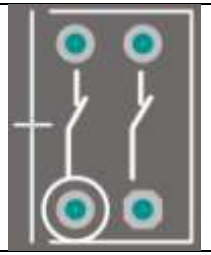
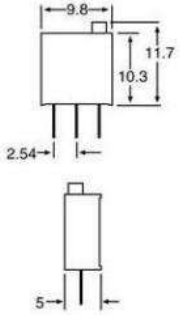
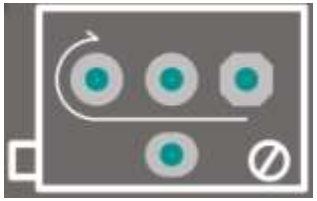
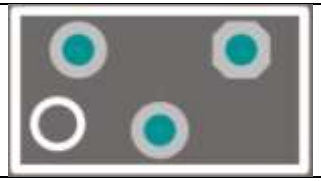
Resistor Metal Film		
Resistor Network		
Connector IDC Boxed Header Power		
Connector IDC Boxed Header Right-Angle		
Connector IDC Receptacle		

ELBY Designs - Laurie Biddulph
9 Follan Close, Kariong, NSW 2250, Australia
elby-designs@bigpond.com http://www.elby-designs.com

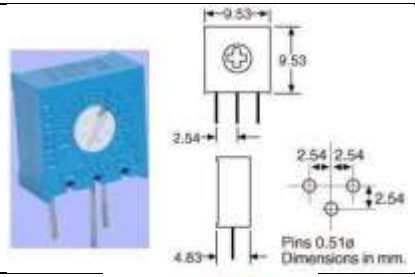
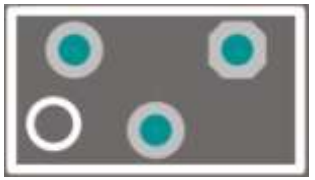
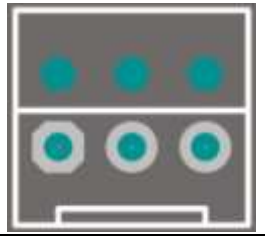
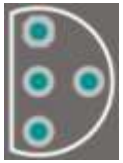
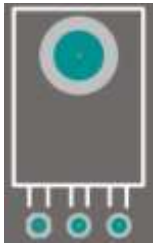
General Construction Notes

Connector IDC Boxed Header Vertical		
Connector MTA Header		
Connector MTA Housing & MTA Crimp	 	
Faston Quick Connect		

General Construction Notes

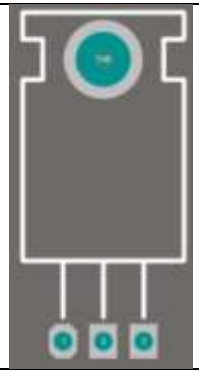
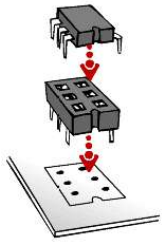
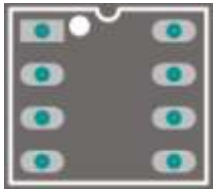
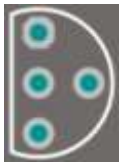
Connector 4mm/Banana Socket		
Connector 3.5mm Mono Socket		
Trimmer 3296W Multi-turn	 	
Trimmer 3306W Single Turn		

General Construction Notes

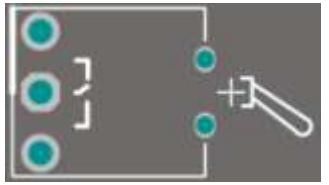
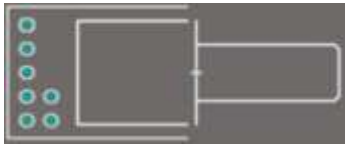
Trimmer 3386H Single Turn		
Connector Screw Terminal Block		
Semiconductor Transistor TO92		
Semiconductor Transistor TO126		

ELBY Designs - Laurie Biddulph
9 Follan Close, Kariong, NSW 2250, Australia
elby-designs@bigpond.com http://www.elby-designs.com

General Construction Notes

Semiconductor Transistor TO220		
Integrated Circuit with socket. (8-pin shown)		
Semiconductor Transistor TO18		

General Construction Notes

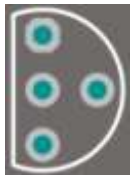
Switch Single-pole, Toggle		
Switch Rotary		

NB: Components titled in red are polarised and must be installed with specific attention to their orientation.

General Construction Notes

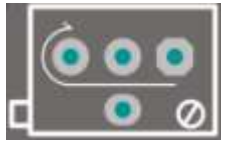
Multi-Footprints

Some footprints are designed to support multiple packages.

For example our standard TO-92  footprint will accept transistors with :-

inline legs  or staggered legs 

When installing these components there is no need to bridge or solder the unused pad.

Similarly, our standard trimpot footprint  will accept :-

inline legs  or staggered legs  this also includes horizontal and vertical trim 

Again, there is no need to bridge or solder the unused pad

Bill Of Materials

The Bill Of Materials (BOM) is a list of all the components required in the assembly. It lists the item by description and where available, the designator for that item on the board.

BOMS for some 3rd-party products such as those for Ken Stones CGS family do not have designators as the board itself has the component value printed on the board. In these cases it is important to refer to the current documentation (web pages in the case of CGS) for the latest notes regarding changes to component values.

Panther modules consist of one or more columns and these are numbered from left to right. The BOM for a Panther module is broken down in to sub-assemblies with each sub-assembly corresponding to a column in the module. The heading for each individual BOM will include the column number in the title at the top of the BOM.

Component Specifications

The BOM will provide, for each part, a description and a package. The package reflects the footprint used on the pcb and so also defines the optimum size/shape of the part for fitting to the board. Other package/size parts may be used but may require to be 'prepared' prior to installation. For example, a 0.3" pitched axial resistor may be easily used in place of the specified 0.4" axial part and will simply require the legs to be formed at the 0.4" spacing. A 0.1" radial capacitor can be used in place of a 0.2" radial capacitor but will require that the legs be formed before installation.

The description for a part, particularly resistors and capacitors will often indicate voltage rating and a wattage rating. Where given, these reflect the 'standard' parts supplied in our kits and does not necessarily stipulate the actual specification for the part. For example, many electrolytics are specified as 50V devices but in most cases these can be rated as low as 16V (most of our designs use 12V rails so 16V is perfectly okay, for 15V supplies we would recommend a minimum of 25V). You could possibly also use larger voltage devices but inevitably they are physically bigger and so there may be issues with component spacing. As another example we specify 0.5W for resistors. In most situations you can safely use 1/4W resistors and sometimes even 1/8W resistors will be okay. Attention should be paid to the location of the substituted component within the schematic to determine if a lower specification part can be used. Resistors in series with power rails will usually require a larger power rating than those used in other parts of the circuit. Capacitors (specifically tantalum but also electrolytics) directly across the power rails should always have a voltage rating greater than the power rail and for

tantalums you should be aiming for as close to triple that voltage i.e. on 12V rails you should have, as a minimum, a 35V device (electrolytics need only be larger than the rail voltage so 16V would be fine for a 12V rail).

Soldering Components

it is important that you have basic soldering skills. There are plenty of good videos on the internet and novices should look at these for guidance and, if necessary, experiment on building some small cheap kits before working on our kits.

Most modern-day components are designed to sustain a high temperature soldering phase for a few seconds without damage to the component. However some components are more temperature sensitive and so you should minimise the time taken when soldering these components. A prime example are 'naked' axial polystyrene capacitors. having very thin wires and a soft body, heat is quickly transferred in to the body which, if exposed to heat for too long, can start to melt.

Surface Mount Components

Some kits need to use SMT components. When needed we always try to use 0805 or larger footprints for resistors, capacitors and LED as these are generally easy for everyone to solder.

With the 0805 (and larger) and SOIC (and larger) SMD components you will not have any real problems. It is just a matter of developing a good technique and having the right tools.

You should get yourself a pair of fine nose tweezers (ones that go to a sharp point and the tips meet cleanly and firmly.

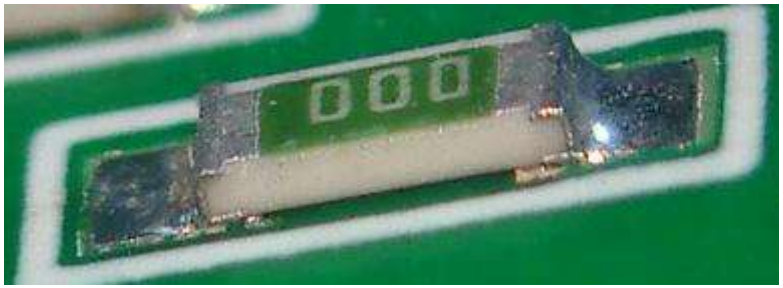
When soldering these components you should use a fine gauge solder (less than 0.7mm is recommended) and a soldering iron with a fine tip.

SMT Resistors and Capacitors

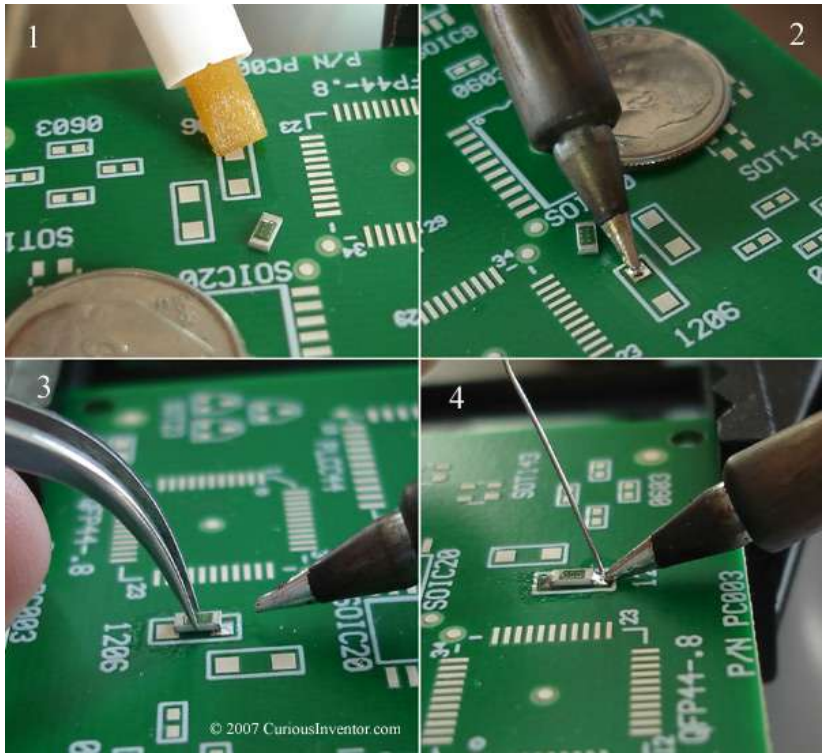
1. Optionally apply some flux to the pads
2. Apply a small dab of solder to one pad on the pcb for the component.
3. Hold the component with a pair of small-nosed tweezers and bring it up to the pcb footprint with the body parallel to the pcb. Apply heat to the soldered pad and when it flows slide the SMT component in to position. Remove the soldering iron, let the solder joint set and then release the tweezers. The component should be sitting reasonably central between the pads and have enough exposed pcb pad on the unsoldered end to allow it to be soldered. Check that the component is flat against the PCB, if not then apply a bit of downwards pressure to the component and then reflow the soldered pad.
4. Put the tip of the soldering iron on the pcb pad to heat it up, apply solder and then move the soldering iron in to contact with the SMT component pad. The solder should flow with the tip and on to the component pad.

Being right-handed, I pre-solder all the right-hand pads on horizontal components and top pads on vertical components and then use the above method to fit all the SMT components.

General Construction Notes



A good solder joint, not too much solder.

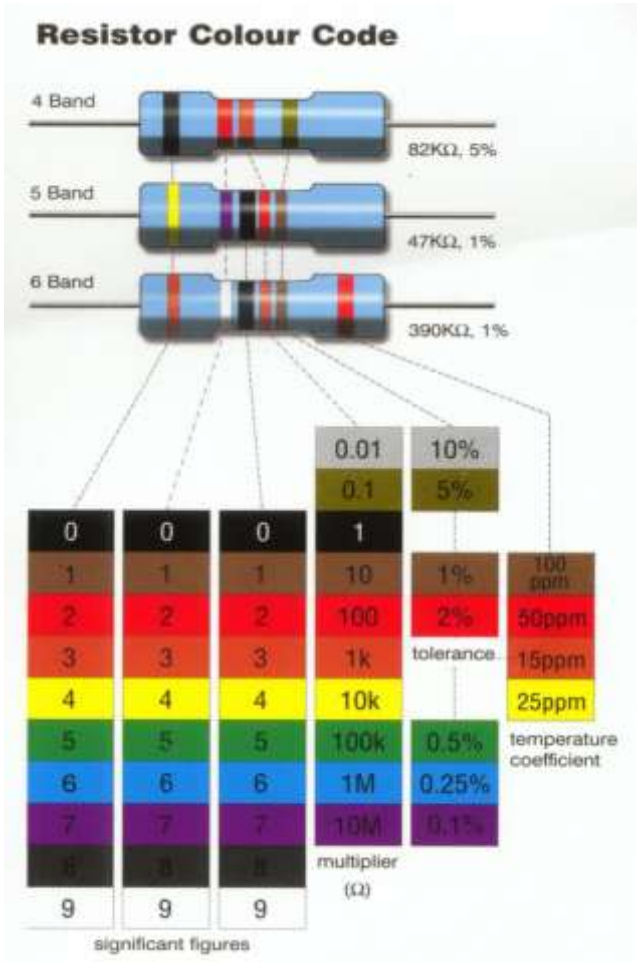


For SOIC devices you again place it on its pads this time making sure that all the legs sit squarely on their pads. Press the centre of the device with the tweezers and solder 2 diagonally opposing pins. Double check that all the legs neatly line up on the pads on the pcb and that they don't bridge to an adjoining pad. When happy, carefully solder the other pins. For small SOIC's like this I would suggest you try soldering each pin separately. Do this by keeping the amount of solder on the iron as small as possible. Clean the iron tip between every solder operation, this will minimize the build up of dirt and solder and allow you to maintain control over how much solder is being applied.

If you do end up with a solder bridge between 2 legs then don't try using the iron to remove the excess, you will inevitably bend the legs, make it worse or cook the chip. Use a piece of solder-wick, place it over the solder bridge and then press the iron tip against the wick, you will see the solder flow in to the wick. Remove the iron and the wick, at the same time, making sure you move AWAY from the legs and not across them.

Once soldering is complete wash the board thoroughly to remove all traces of flux. Use a toothbrush and a suitable flux cleaner.

Resistor Colour Codes



Real-time online calculator can be found here [Online Resistor Calculator]

General Notes - Problems

The most common error with pcb assembly is inserting parts into the wrong holes or incorrectly orientated (polarized components only). Please double check every part before you solder any part into place. Desoldering parts on a double-sided board is a skill that takes a while to master properly and, if not done properly can not only damage the component but can also damage the pcb.

If you have put a component in the wrong place, then use wick or a good solder pump to remove the solder from the hole. Resoldering the joint with fresh solder can actually make the solder easier to remove. Once removed, ensure the hole is clear of solder,

Make sure your pcb is free of grease and dirt before you start soldering. An alcohol based cleaner like iso-propanol is a good choice as this will dissolve the grease and dry off quickly. I also use this to clean the board during assembly to remove excess flux that builds up through the soldering process.

You should ensure that your soldering iron tip is at the correct temperature as recommended for the solder being used. Too cool and you will not get the solder to flow properly and will have poor and unreliable solder joints. Too hot and the flux in the solder will burn off too quickly preventing a good solder joint from forming properly and you run the risk of damaging the component. This is particularly true of soft-bodied components like polystyrene capacitors and pcb-mount connectors with plastic bodies.

All resistors should be flat against the board surface before soldering. It is a good idea to use a ‘lead bender’ or pair of snipe-nosed pliers to pre-form the leads before putting them into their places. Try to keep the bend in the leg away from the point where the leg meets the component body as this may stress the joint. Once the part is in its holes, bend the leads that stick out the bottom outwards slightly to hold the part in place. This is called ‘cinching’. This only needs to be a slight bend as the intention is to prevent the component from falling out when the board is turned over for soldering. Also, the slight bend means that should you need to remove the component then it will be a lot easier than if the leg has been bent at 90°.

Turn the board over and solder from the bottom of the board, applying the solder so that the hole is filled with enough solder to make a small cone around the wire lead. Don’t put too much solder on, and don’t put too little on either. Clip the leads off with a pair of side cutters, trim level with the top of the little cone of solder.

Once all the resistors have been soldered, check them ALL again. Make sure they are all soldered and make sure the right values are in the right place.

General Construction Notes

IC sockets are recommended, especially if this is your first electronics project. We recommend the use of turned-pin sockets as these offer a superior `grip' on the ic legs and thus better electrical connectivity.

General Construction Notes

3D Models

To aid with construction of our kits we are introducing the use of 3D Models in to our build documentation. Each build document will have a link to one or more 3D PDF(*) file(s) that show the particular board indicated (**).

For each model you can select a component and hide or show it using the tickbox list on the left. This will allow you to quickly identify any component and determine its location and orientation on the board as well as visually identify a components physical appearance (***).

The 3D model can be rotated in all axis by holding down the left-mouse button and moving the mouse, and zoomed in or out by holding the right-mouse button.

The use of these 3D models is felt to offer many advantages to the builder over the more common approach of showing images of boards at various build stages. The 3D model allows you to build in any order you desire - you simply hide all the 'other' components. You can save copies of these intermediate stages for future reference if desired.

See ReadMe for more information on 3D PDF

General Construction Notes

Linux Users

If not already installed, you will need to install Acrobat Reader 9.5 or later:-

sudo dpkg -i AdbeRdr9.5.5-1_i386linux_enu.deb

(manually installs the package AdbeRdr9.5.5-1_i386linux_enu.deb - which will have to be obtained by user)

sudo apt-get install gtk2-engines-murrine:i386 libcanberra-gtk-module:i386 libatk-adaptor:i386 libgail-common:i386

(obtains libraries that are somewhat deprecated and no longer necessarily included with current distribution)

sudo apt-get -f install

(collaborates the packages and dependencies which may have conflicted previously)

(*) requires the use of a 3D capable PDF viewer (Acrobat Reader 9.5 or later). You may also need to activate the 3D viewer.
(**) All 3D documents are stored in a separate folder from the build notes. You will need to have internet access to view these documents.
(***) Markings, colour and, in some cases, size of components in these 3D models is purely indicative of the component and do not show the actual component specific details i.e. all 0.5W 1% Metal Film resistors are shown with the same body colour and markings

EuroSynth Specification

ELBY Designs is now adopting the EuroSynth Specification which defines specifications for various criteria of a EuroRack system that, when implemented, ensure a coherency across the system.

As some of these specifications impact on the current designs of most modules, some of the specifications may require time to filter through to production level.

Specifically this relates to the adoption of a separate CHASSIS GROUND from the existing 0V rail to allow proper grounding and isolation methods to be employed.

In addition, the EuroSynth Specification has removed the CV, GATE and +5V signals from the power connector and will be using a 10-pin IDC boxed header at both the module and busboard ends. With regards the module end, the IDC-10 header will be implemented immediately and will have no impact on existing modules including those from other manufacturers and who already use a 10-pin connector.

For modules using a 16-pin connector but not requiring +5V, CV or GATE, all is required is that they start fitting a 10-pin IDC on to the existing footprint.

During the transition period with modules, we will now be supplying 10-pin IDC headers in place of the original specified 16-pin Header. When mounting 10-pin headers these on to the 16-pin footprint, simply install with pin 1 of the 10-pin header aligning with pin 1 of the 16-pin footprint.

Busboards will continue to be supplied with 16-pin headers for the short-term future and will so require a 16-10 pin cable for module power. In the future, modules requiring access to the CV/GATE will need a special cable to access a dedicated 'DATA BUS' which is yet to be defined. Modules requiring access to the external 5V rail will need to use an inline adaptor which will generate +5V from the +12V rail.

Module Power Connections

It is imperative that a module is connector to the power system correctly to prevent damage to the module and/or power supply. EuroSynth modules used boxed headers along with electrically and mechanically correct power cables, throughout the system which eliminates the possibility of connectors being misaligned or reversed.

As per the EuroSynth specification Pin 1 = -12V. On IDC cables, pin 1 is also identified by a red stripe.

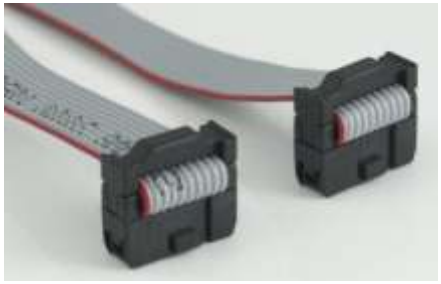


Figure 37b

A mechanically and electrically correct IDC cable is shown at left:-

1. There is a marker on the IDC socket (usually a TRIANGLE) at the pin 1 position - figure 37c
2. There is a similar marker on the IDC header pointing to pin 1 - figure 37b
3. The IDC socket has a polarising bump on the same side as pin 1 - figure 37c
4. The IDC header has a mating polarising notch on the same side as pin 1 - figure 37b

Figure 37c