

Electronics for Model Railways



Chapter 19

Construction methods

By Davy Dick

Electronics for Model Railways

By Davy Dick

© 2020 by David Dick

All rights reserved under the Attribution-Non-Commercial-NoDerivatives Licence.

This book may be freely copied and distributed but may not be changed or added to without prior written permission of the author.

This book is free and its material may not be used for commercial purposes.

This book is issued as, without any warranty of any kind, either express or implied, respecting the contents of this book, including but not limited to implied warranties for the book's quality, performance, or fitness for any particular purpose.

Neither the author or distributors shall be liable to the reader or any person or entity with respect to any liability, loss or damage caused or alleged to be caused directly or indirectly by this book.

All trade names and product names are the property of their owners.

In memory of Margaret



Contents

- Chapter 1 - Basic Electronics
- Chapter 2 - Loco controllers
- Chapter 3 - Layout wiring
- Chapter 4 - Track wiring
- Chapter 5 - Voltage problems
- Chapter 6 - Point wiring
- Chapter 7 - Point motors & servos
- Chapter 8 - Scenic lighting
- Chapter 9 - Coach lighting
- Chapter 10 - Adding sound
- Chapter 11 - Track occupancy detectors
- Chapter 12 - RFID
- Chapter 13 - Digital Command Control
- Chapter 14 - CBUS
- Chapter 15 - EzyBus
- Chapter 16 - Arduinos
- Chapter 17 - Breadboarding
- Chapter 18 - 3D printing
- Chapter 19 - Power supplies & cutouts
- Chapter 20 - Transistors, ICs and PICs
- Chapter 21 - Interfacing techniques
- Chapter 22 - Computers and model railways
- Chapter 23 - Assembling a tool kit
- Chapter 24 - Soldering
- Chapter 25 - Using test equipment
- Chapter 26 - Pocket Money Projects
- Chapter 27 - Abbreviations & Acronyms
- Appendix - The Model Electronic Railway Group

Construction methods

When you start off in this hobby, you are content to use pre-built commercial equipment.

After a while, you may venture into building modules from kits.

Hopefully, you will learn some more along the way.

Eventually, you may wonder if you can alter or add to an existing kit.

You may decide to attempt to build a module from a circuit diagram you saw in a magazine or on the Internet.

You may even decide to design your own circuit.

This chapter considers the different methods of creating and developing circuits.

It depends on whether you are creating a temporary test bed or a permanent module.

Temporary construction

It may be as simple as making up a quick test bed to check the performance of a component or module you just acquired.

You may be using a test bed to check if a component is faulty.

You may be checking if you can modify an existing circuit (e.g. can you change the timing, or can you replace that LED with a relay).

All of these will require some trial and error.

You rarely get things right first time round.

In these circumstances, you need the ability to rapidly assemble components and test a circuit. You also need to be able to rapidly change components or move components.

What you need is a method that allows flexibility without too much inconvenience, maybe even without the need for soldering and de-soldering components.

Common temporary methods are:

- Open wiring or 'Dead bugs'
- Breadboards

Semi-permanent or permanent construction

The intention is to produce a module that can be used after it is completed.

- It may be something that is not available commercially.
- It may be creating a module that is cheaper or has more facilities than a commercial version.
- It may be converting an old design from transistors to ICs or PIC chips.

You are likely to have used a breadboard or similar to confirm the circuit meets your needs and are now looking to make a more durable version.

Common methods here are:

- Stripboard
- Perfboard
- Printed circuit boards

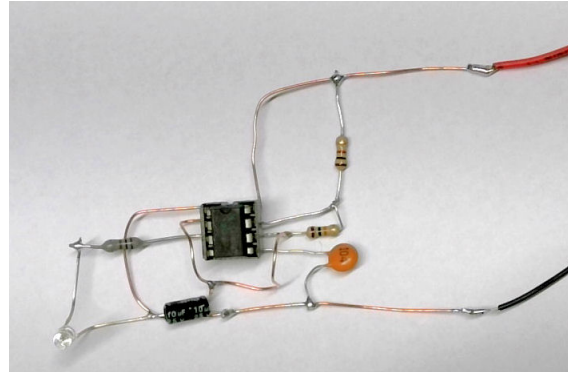
Open wiring

For a quick and dirty mock-up you can just solder all the components together.

Its not pretty but it is quick.

It does not involve anything other than the components and maybe some extra wires.

You can make the layout closely resemble the actual circuit diagram, which helps avoid mistakes.

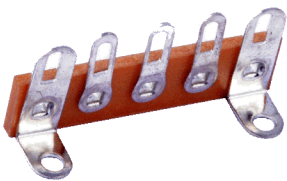


Alternatively, with great care, you can pack the components tightly together into a block and then cover using a glue gun or a resin. This need a good eye and considerable soldering skills but can result in a permanent assembly. With surface mount components, this is an even greater challenge.

Although crude, this can sometime result in a very compact module as it does not need the area of a mounting board but it does need to be done carefully.

However, open wiring poses more of a problem with a larger circuit.

It becomes unwieldy due to lack of mechanical support, it is more prone to unwanted shorts, and it occupies a large amount of space.



To get round this, some enthusiasts solder components and wire on to tag strips. These come in various lengths and provide more support.

However, the layout can no longer mimic the circuit diagram, with increased chance of mistakes.

A variation on this is the '*dead bug*' technique.

This is often used where there are several ICs in a module.

ICs are placed (sometimes glued) on a board with their legs sticking upwards (hence the title '*dead bug*'). The components are then soldered to the legs of the ICs.

These techniques are handy when no other method is available, or where space is an absolute requirement.

Breadboard

All other methods in this chapter result in either permanent or semi-permanent circuits.

Breadboards are different. They are used for creating temporary circuits.

In the early days, hobbyists hammered nails into a wooden board that was used in the kitchen for slicing bread. Components and connecting wires were then soldered on to the nails. The method has moved on but the name has stuck.

A modern breadboard is a plastic box with holes into which you can insert the wire legs from LEDs, resistors, capacitors, transistors, wires – and anything that has a 2.54mm (0.1”) pin spacing (integrated circuits, trimmers, terminals, etc.).

For those just starting, there is nothing more off-putting than building a kit and finding it does not work. In most cases, this is a result of poor soldered joints or solder shorting to places it ought not.

With breadboards, you simply poke the electronic components into its holes and connect some of them together with wires. The result is more likely to be a working module – and with no soldering whatsoever. After this learning experience and confidence boost, you can move on to soldering the components on to printed circuit boards or stripboard.

For more experienced members, breadboards offer the opportunity to experiment, without constantly soldering and unsoldering components. Components can be inserted and replaced easily without any damage. This is much quicker than soldering/unsoldering. Since this is a temporary setup, both the breadboard and the components can be used over and over again.

Possible uses for breadboards include:

- Learning – e.g. how a chip or a transistor works.
- Experimenting – e.g. what is the best resistor value, or possible modifications to an existing circuit.
- Testing – trying out electronic circuits you found in magazines or on the internet. You can confirm that it is what you want before constructing a permanent version.
- Prototyping – having a go at designing your own circuit and being able to quickly test and alter it.
- Demonstration – making up a temporary assembly to demonstrate the operation of a particular circuit or component.

The main benefits of breadboards are

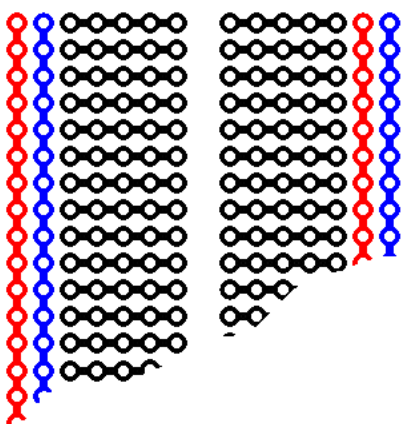
- Circuits can built quickly
- Alterations can be made quickly.
- They can be used over and over again for different circuits.

How it works

This image shows a typical small breadboard that is widely available.

They are available in a range of other sizes and configurations.

The one shown here has 30 rows and 4 columns.

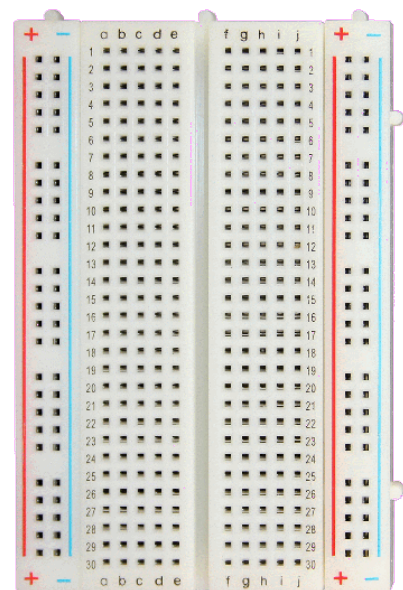


This illustration shows what is happening inside the breadboard.

The holes in the vertical columns are all connected together internally and are used to distribute the power along the breadboard. These are the '*bus strips*' and for convenience they are marked as + and – and have red and blue (or sometimes black) stripes next to them.

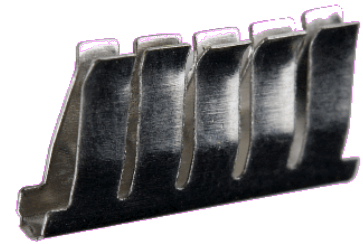
blue (or sometimes black) stripes next to them.

The horizontal rows are known as '*socket strips*' and they are connected together internally in sets of five, facing further sets of five with a gap in between.



This gap allows integrated circuit chips to be inserted across the gap, with legs connected to adjacent rows of holes. The gap prevents shorting between the rows of the chips' pins.

This image shows the metal sprung contacts that are beneath the breadboard holes.



The breadboard often has its rows and columns numbered to aid clarity and reduce mistakes when inserting components/wires.

In our example board, the holes 'a' to 'e' in row '1' are commoned together, so up to five components or wires can be connected together.

Powering the board

Some of the more expensive breadboards come with power supplies fitted to them.

The more basic versions require an external power supply.

If required, you can plug wires from any power supply into the power rails of breadboard.

So, if you are working on a 12V circuit, you would connect 12V DC to the breadboard

power rails; if working with 5V devices, you would connect a 5V supply to the power rails.

Also, for around £1, you can buy a breadboard supply module as shown in this image.

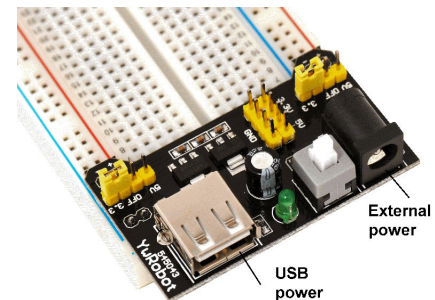
The module plugs straight into power rails of the breadboard and provides two options for connecting power. There is a socket that connects to any

external DC supply between 6.5V to 12V. This could be a bench supply, a wall wart, a power brick, or a 12V DC outlet on your controller.

Another option is to connect the module to a USB socket on your computer.

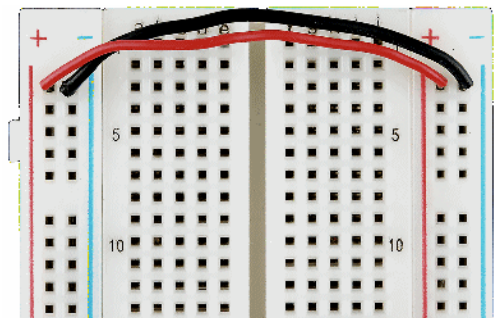
This module provides an off/on switch and a power indicator LED.

It can provide either 3.3V or 5V to breadboard rails (or even 3.3V to one set of rails and 5V to the other set).



Note

The power rails on the left and right hand sides of the breadboard are not normally connected together internally. If you need to have the same voltage available on both sets of power rails, you can link them together with jumper wires.



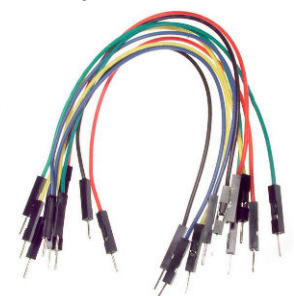
Wiring things together

As mentioned, each socket strip has five holes that are connected internally, so no more than five leads can be inserted in any one socket strip. This is not a restriction as you can use wires to connect socket strips together.

So, for example linking two strips allows eight leads to interconnect (you have lost two because of the link).

You will also wish to connect components that may be inserted in different places on the board.

While you could use lengths of wire, it may be more convenient to buy the male-to-male breadboard jumpers as shown in this image.



A set of jumpers is quite cheap and they have miniature prods on their ends that are designed to provide a snug fit (i.e. good contact) when inserted in the board. Sets are supplied with different colours which can be helpful for colour-coding. For example, I use red wires for jumpers that carry 12V and orange jumpers for carrying 5V, with black jumpers for 0V.

If you do not have any jumper leads to hand, you can utilise solid wire (avoid using multi-strand wire).

Breadboarding allows for great flexibility in layout and wiring.

The component layout on the breadboard need bear no similarity to the circuit diagram – as long as all the components and jumpers replicate the interconnections shown in the diagram. You can quickly move components around the breadboard to suit yourself. Just minimise jumpers crossing over each other and ensure they are clearly visible.

Connecting external devices

You may want to develop circuits that have components that will not easily fit on the breadboard. Examples could include motors, servos, LED strips, speakers, etc.

Terminal blocks with 2.54mm pin spacing can be plugged into the breadboard and external devices can then be connected to the electronics on the breadboard.

Examples

There is no 'correct' way to use breadboards but here are a few examples that might show their use.

LED and dropper

Here is the simplest example that tests a LED.

You can use it to check whether a LED is working.

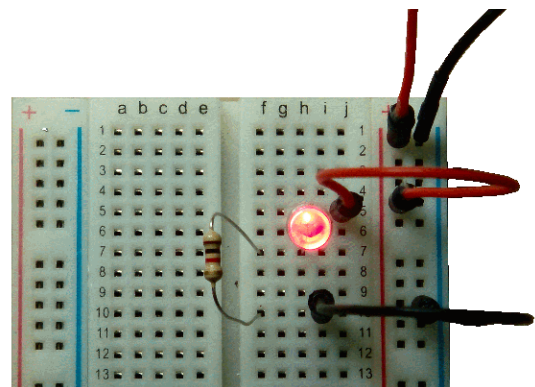
You can also check which end is the anode in situations where the LED has its leads cropped to the same length (e.g. from LEDs recovered from soldered boards).

The red and black leads leaving the board are connecting to a power supply.

A 1k resistor works with most LEDs.

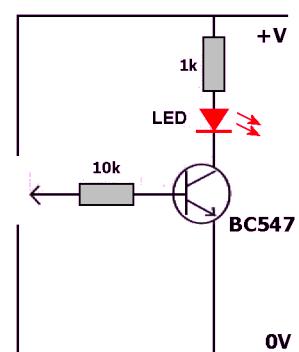
You simply plug the LED into the board – in this case between row 5 column h and row 7 column h.

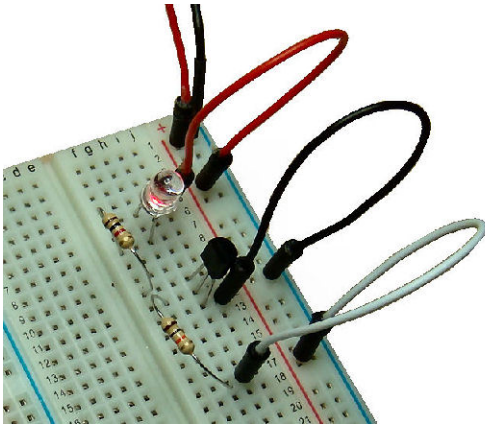
- If the LED lights, it works.
- If the LED does not light but lights when reversed, you have found the LED's anode (the lead nearest the red jumper).
- If the LED does not light either way round, it is kaput.



A transistor switch

Here is a simple circuit that shows how a BC547 transistor (an n-p-n type) operates. The transistor will switch the LED on when the 10k resistor is wired to the positive rail and will be extinguished when then the 10k resistor is wired to the 0V rail.





Here is the circuit's implementation on a breadboard. It also uses a 1k resistor as it works with a range of LEDs and power supply voltages. The 10k resistor prevents the transistor from drawing excessive current and being destroyed.

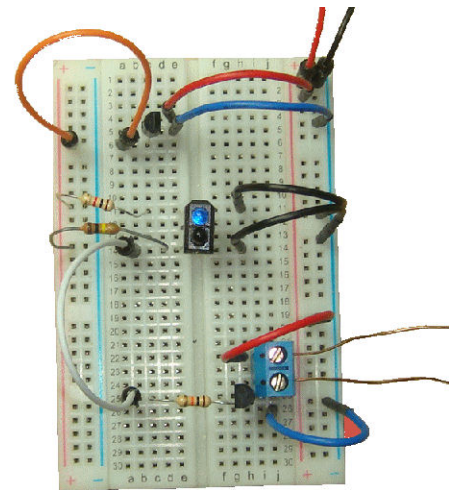
Something a little bigger

You can see that this circuit uses three distinct sections. The wires coming in at the top of the breadboard are connecting 12V to the right hand power bus strips of the board. The 12V is taken over to a voltage regulator and its 5V output (note the orange jumper) is used to feed the left-hand power bus strip. In this way, we have created both a 5V supply and a 12V supply, with a common 0V. The 5V is used to power an infrared detector and its output (the white jumper) is taken down to a transistor switch (like the one described above).

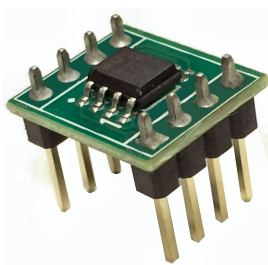
The output of the transistor is taken to an external 12V LED strip via the 2-pin connector block.

Note that the infrared detector straddles both rows.

Standard ICs in DIP (dual in-line package) format can also straddle across two rows, as in this image:



You cannot insert a surface mount IC on to the board (its legs are too short!).



In these circumstances, you can buy an adapter / breakout board. You then solder the surface mount IC on to the breakout board, as shown in this image.

They can then be inserted to straddle the left and right hand rows of contact in a similar fashion to the standard IC shown above.

Limitations

Although you can buy boards with more holes and can connect multiple boards together, very large circuits can make the whole process overly complex and unmanageable. The components are only connected through friction instead of permanent soldering. Therefore, breadboards are not suitable for projects that have excessive voltages and/or currents. The limits depend on the quality of the breadboard but staying within 50V and 500mA should be perfectly safe – and this covers most modules you will be playing around with. Other restrictions include working with surface mount devices (unless you solder them to breakout boards first) and unwanted stray capacitive and inductive effects at high frequencies (not found too often with model railway projects).

Jumper wires rely on being held tightly by the sprung contacts and the efficiency of that depends on the quality of the breadboard.

Poor connections lead to unwanted resistance or to intermittent working.

Loose connections allow the jumper to easily fall out or be accidentally pulled out.

These problems may increase after many uses of the breadboard. Fortunately, they are relatively inexpensive.

Software for breadboarding

For a simple circuit you can lay out the components with little regard to placement.

For more complex circuits you can use software to make the task easier.

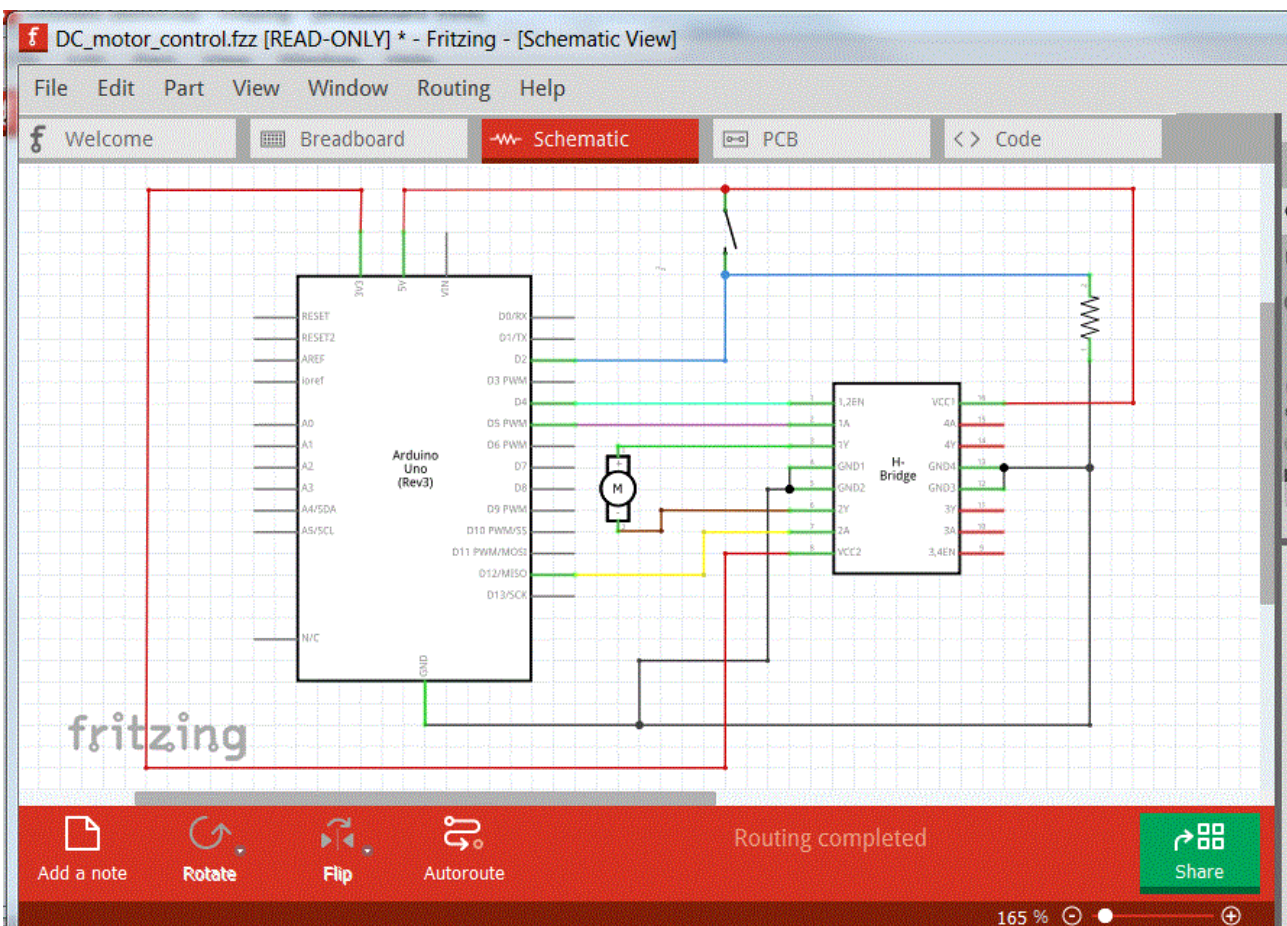
One package of note is Fritzing, a comprehensive layout application that is available at:

<https://fritzing.org/home/>

Fritzing allows you pull components on to the screen and make connections between them.

Once you complete your circuit, you click on the '*Breadboard*' tab to reveal a suggested wiring diagram.

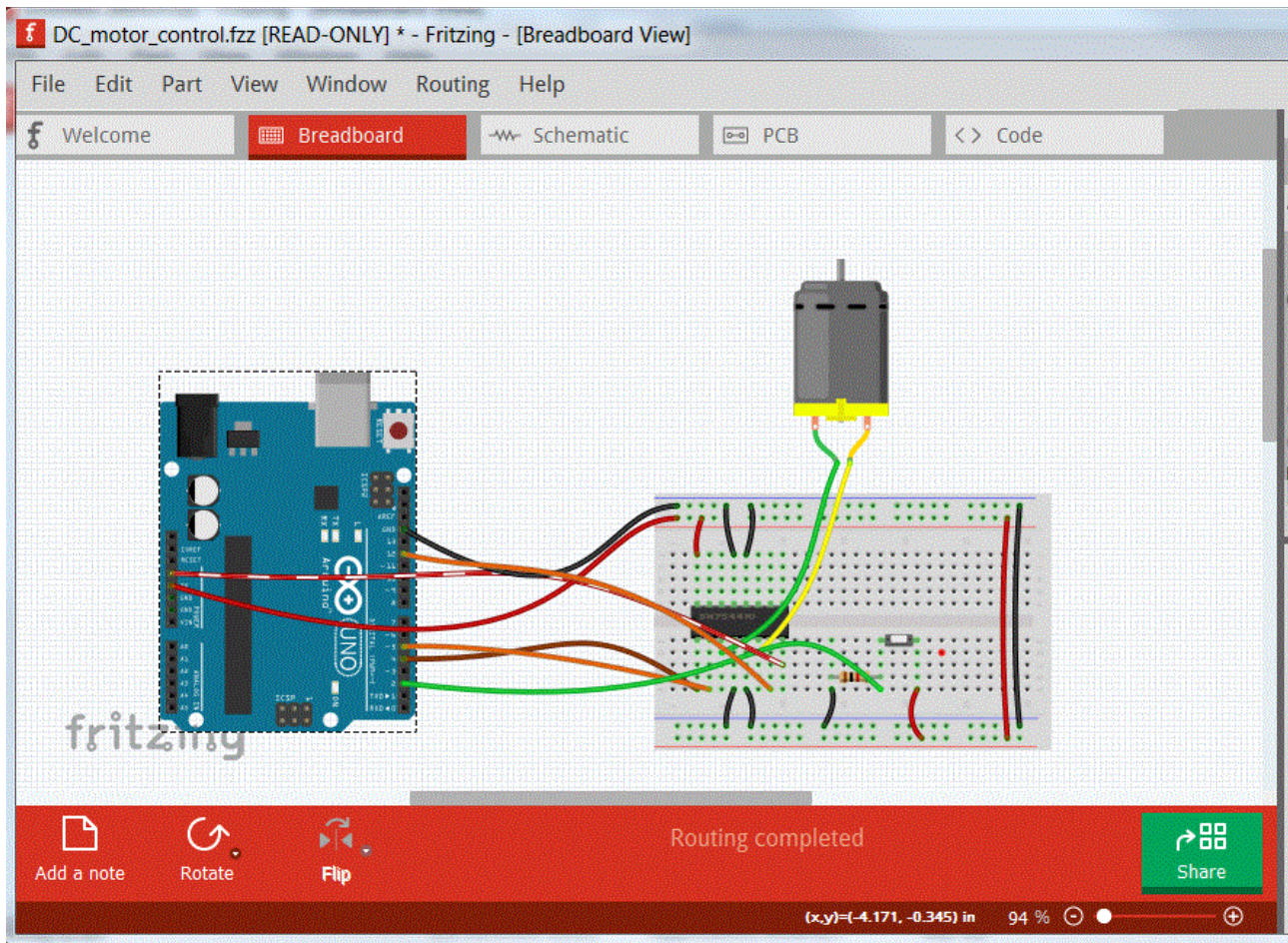
Here is an example of a circuit that uses a H-bridge driver to operate a motor under the control of an Arduino Uno.



It has been created in the '*Schematic*' window.

It is up to you to check whether you have used the correct components or wired them together correctly.

Here is the suggested layout that it produces:



You then just have to place the components and add the wiring as shown in the layout illustration.

Stripboard

For the great majority of projects, getting the components is easy; they can be purchased from suppliers such as Rapid, RadioSpares, CPC, Bitsbox, etc., or even cheaper on eBay. Since they are not complete kits, there will not be a printed circuit board available. PCBs are great; they tell you where to put your components and make life simpler for the constructor.

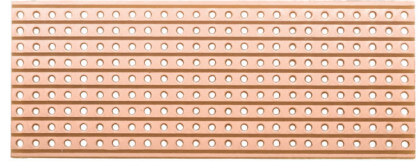
Without a PCB, you need another base on which to build your project. That is where stripboard comes in.

Stripboard (originally called 'Veroboard') is designed as a prototyping board for building a one-off project. The finished project, if proven to be successful, can be put into long-term use or – if it becomes popular - PCBs might be produced for wider use.

Stripboards are relatively cheap and are readily available.

Most stripboards use a thin sheet made from SRBP (synthetic resin-bonded paper - a paper based laminate). Higher quality versions are made from FR-4 (a fibreglass-reinforced epoxy laminate).

As the picture shows, one side has copper tracks bonded to it. The whole board is then drilled through with holes that are spaced 0.1" (2.54mm) apart. This is designed to have the same spacing as ICs, their holders, etc..



Sizes

The most commonly used sizes by hobbyists are

25mm x 64mm - This board has 9 tracks, each track with 25 holes

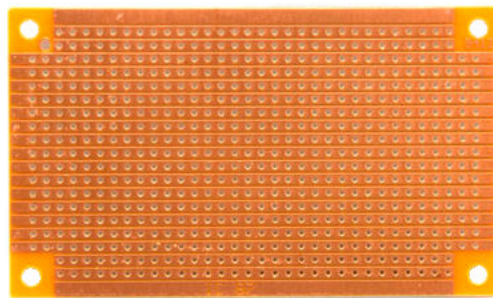
64mm x 95mm - This board has 24 tracks, each track with 37 holes

The 25mm x 64mm version is used in the Pocket Money Projects.

Vero Technologies make a large range of other sizes and details are on their website at:

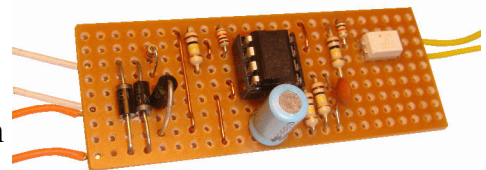
www.verotl.com/circuitboards/veroboards

If required, the boards can easily be cut to size with a small hacksaw.



All the circuit's components are laid on the opposite side to the copper tracks, as shown in the picture.

The wires from the components are bent, then inserted through the holes and soldered on to the copper tracks on the other side.



The copper tracks are used to make the connections between the various components. If you see a picture of a project on a stripboard, it usually has the strips underneath running horizontally.

Cutting the tracks

To use the copper tracks to maximum efficiency, the lengths are often cut, to create smaller runs of connecting tracks.

In the case of the chip shown in the above picture, the track between the rows of chip pins would have to be cut; otherwise the pins would be shorted together.

You could cut the track with a sharp knife but that can be a messy – and dangerous – task. If you are considering making a number of projects on stripboard, it might be worth investing in a stripboard spot face cutter, which quickly does a nice clean job of removing the unwanted copper. The cutter is placed in the desired hole in the copper track and rotated back and forth until a circular area of copper is removed. Check as you go, otherwise you might drive the cutter right through the board!



An alternative to a spot face cutter is a drill bit (3mm or 5mm). Turn the bit by hand; an electric drill would rip a large hole in the stripboard in two seconds flat!

In either case, before moving on, check with a magnifying glass that there are no remaining tiny strips of copper still bridging any of the track sections. Next, clean the copper strips prior to soldering the components.

Positioning

Getting the components into the correct positions and through the correct holes is a tricky business, as it is very easy to make mistakes. After all, even the smallest board has 225 holes (9 rows of 25).

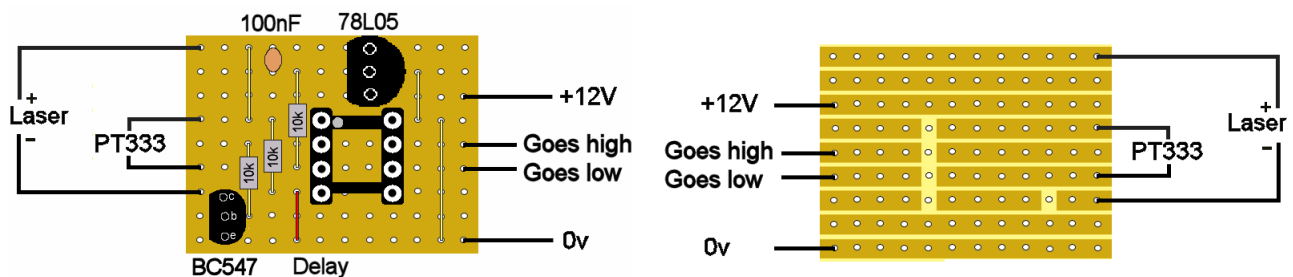
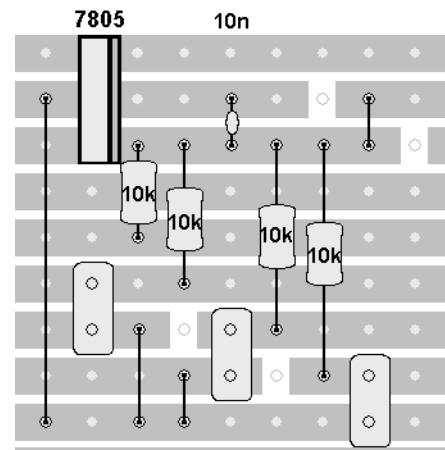
Unfortunately, you can't look at both sides of the board at the same time.

To help, some illustrations show the components and the track at the same time, as in this example. It is really showing where the components should go. It shows the tracks underneath, as if the board was transparent, to help you cut the tracks in the correct places and position the components correctly.

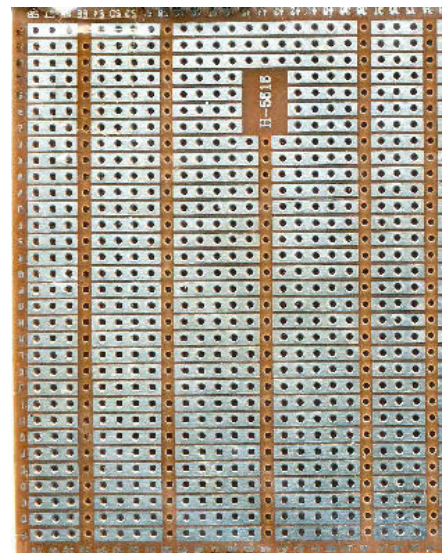
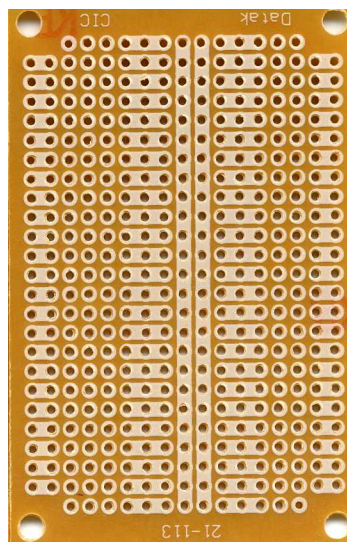
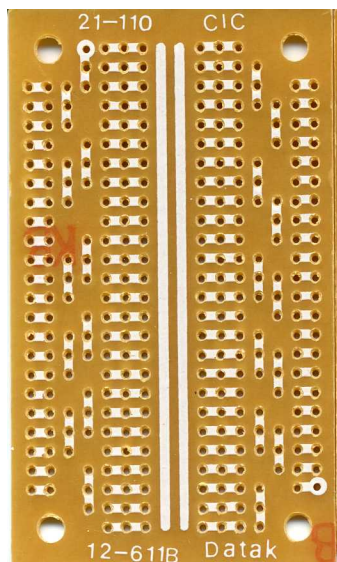
A combination of track cuts and wire links maximises the use of the board.

Other illustrations show the copper side of the board, with any track cuts that need to be made.

The other illustration is of the component side and it shows where to place each component.



Here are some other stripboards/perfboards that offer various types of track lengths.

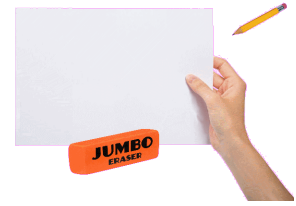


The final soldered module is quite robust and can be used as a permanent module or as a prototype for creating a PCB version.

Software for breadboarding

If you are not comfortable using computers, you can always use a pencil and a sheet of paper (don't forget the rubber!) to plan your component layout, track cuts and wire links.

This can be tedious as you will probably have to go through a number of attempts before you reach the optimum placement. An alternative is to use this planning sheet from Meccano.



ELECTRONICS IN MECCANO	
Stripboard Layout Planning Sheet	
Project:	
Designed by:	
Version:	
Date:	
Notes:	

Actual size stripboard. Hole spacing 0.1" (2.54mm)

Tracks run this way

Tips:

- Mark out the Vs and 0V power lines first, then place the ICs.
- Remember to cut the track between the pins of an IC. Mark the cuts on the diagram with an X.
- Try to make resistors and axial capacitors lay flat on the stripboard. Resistors usually require a gap of 4 holes, capacitors a gap of 8 holes.
- Use the actual size grid on the left to check component spacing.
- Number the pins of the ICs as shown.

Version 3 : 3/02 © 2002 Electronics in Meccano www.eleimec.com

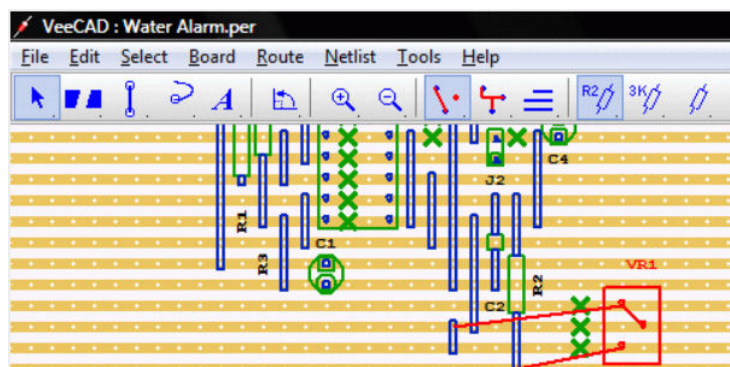
While this planning sheet makes planning a little easier, you will still probably use up multiple sheets until you reach a layout that you choose to use.

Luckily, there are a number of software applications to simplify and speed up the planning process.

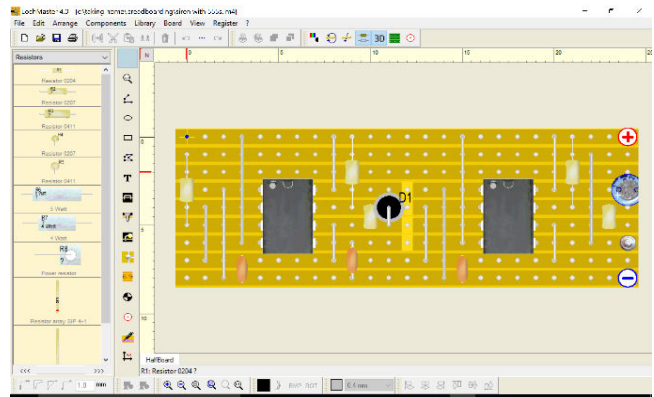
There are a number of older freeware applications such as StripMagic, DIY Layout Creator and VeeCAD.

This is a typical screen from the VeeCAD stripboard designer package.

<https://veecad.com/>



And, of course, the Fritzing application can also be used to design stripboard layouts.

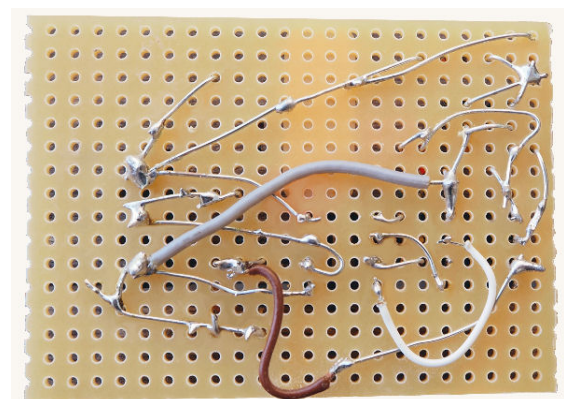
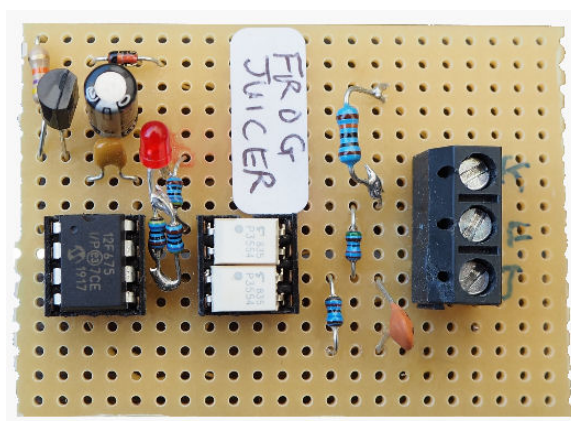


Like stripboard, perfboard consists of a fiberglass board with a 0.1" grid of holes. In most boards, none of the holes have an electrical connection with any other hole. Some boards (see earlier) are a hybrid between stripboard and perfboard in that they have some short strips that connect between some holes in the rows and columns of holes; this is designed to reduce the number links and jumpers that have to be used.

The legs of transistors, resistors, etc. are poked through the holes and soldered to the pads. This provides a more secure mechanical connection.

The finished construction can be used as a permanent module as it is rigid and durable.

Here is an example of a project being developed on perfboard by MERG member Julian Coles.

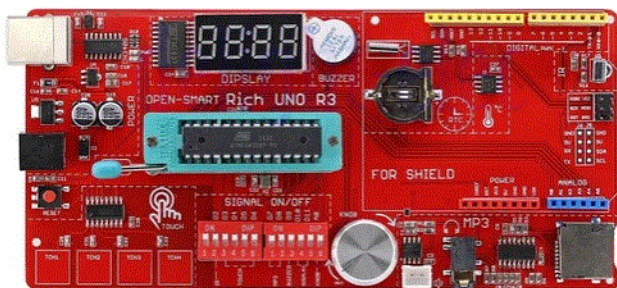
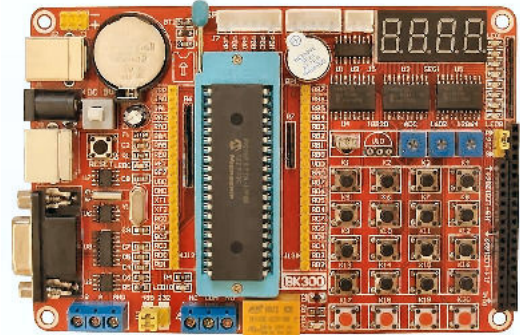


Development Boards

If you are looking to play with PIC chips or Arduinos, you can use any of the above methods.

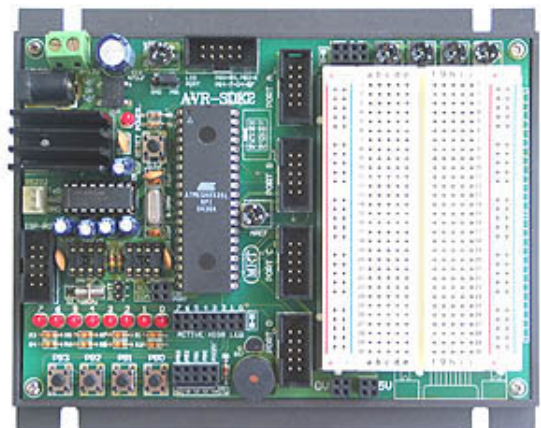
If you are willing to pay a little extra, you can buy ready-built development boards that provide facilities such as pushbuttons, switches, LEDs, LED displays, sounders, potentiometers, etc.

Here is a development board for working with PIC chips.



The above boards have sockets for connecting external devices.

This development board incorporates a breadboard so that you can have the benefit of both.



Here are some suppliers of development boards:

www.futurlec.com/DevelopmentBoards.shtml

www.sparkfun.com/categories/393

<https://cpc.farnell.com>

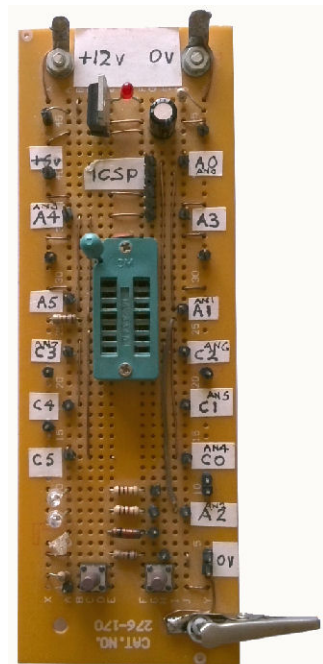
You can always make up your own development board.

This image shows a basic board with a ZIF (Zero Insertion Force) socket for a PIC chip and an ICSP socket to program the chip.

It is powered by 12V and has its own 5V regulator.

All the PIC's pins are taken out to pins down the side of the board.

The board also includes a couple of push buttons and a few LEDs. Using jumper leads, some basic input/output testing can be carried out. This could be checking that the code being written acts on inputs and changes outputs.

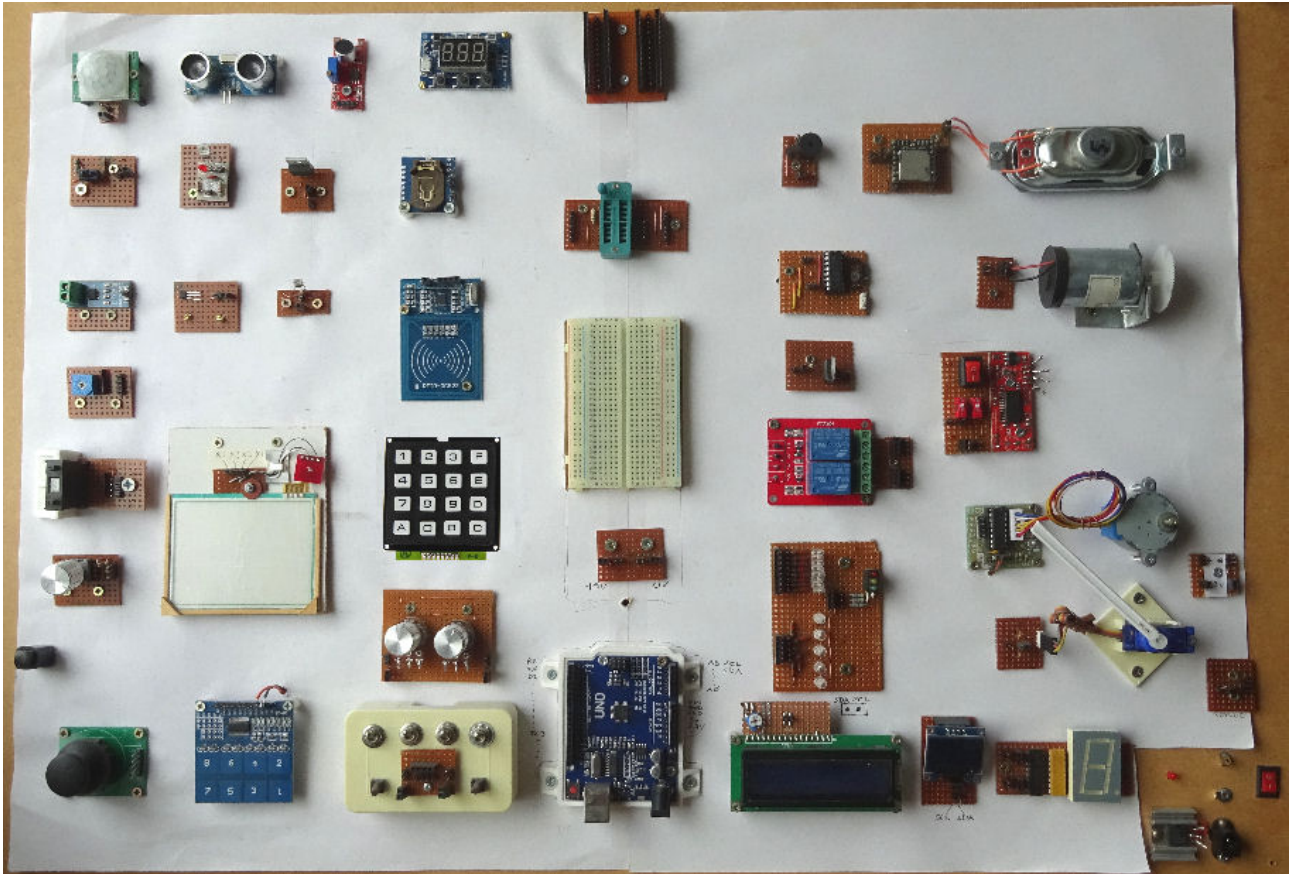


This board has sockets for both PIC chips and Arduino Nanos, so that either can be used for developing a project.

It includes switches, pushbuttons, LEDs, a reed relay, a Sound Player with speaker, an OLED screen, a sounder and two stepper drivers.

The inclusion of a breadboard allows other components or devices to be added.

Connections between the various input and output devices can quickly be made using jumper leads.



This larger DIY development board has multiple input devices on the left hand side of the board (switches, keypad, rotary encoder, pushwheel BCD switch, touch pads, pots, Hall Effect switch, joystick).

It also has multiple sensors (infrared, ultrasonic, RFID, LDR, current, temperature).

Down the right hand side are all the output devices (LEDs, relays, sounder, SoundPlayer and speaker, motor, servo, stepper motor, LCD screen, OLED screen).

In the middle is a breadboard and provision for both PICs and Arduinos.

Printed Circuit Boards

Printed circuits are in use everywhere. You have them in your domestic appliances, your car, your computer and your model railway equipment.

They have a more professional look than the previous methods and are intended for creating multiple copies from a master design.

A single-sided PCB consists of a non-conducting substrate with a copper coating. Parts of the coating are selectively removed leaving conductive tracks and pads.



Another common format is double-sided, with copper on both sides.

This provides great flexibility in the component layout, with connections between both layers achieved by having copper lining though holes in identical locations on both sides of the board (known as 'vias').

Single and double-sided PCBs are within the reach of hobbyists. Professionals design multi-layered PCBs, with copper layers sandwiched between substrate layers.

Unlike earlier methods, PCBs easily accommodate the use tiny surface mount components. With standard components, their legs are poked through holes in the copper pads and soldered.

With surface mount components, their shorter legs are soldered directly to the pads.

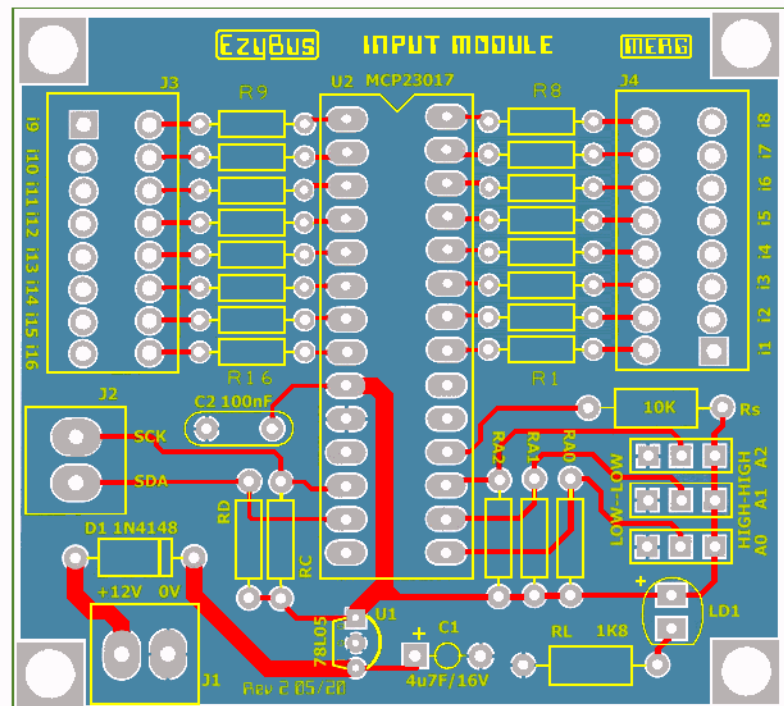
The result is a tidy, compact and reliable module (think of the tiny boards in servos and in DCC decoders)

There is little wasted space on a PCB as the designer can optimise the layout for the best component placement, removing the need for connecting wires/links. The only wires are external ones to and from devices that will connect to the board. The design can aim for maximum compression or for best clarity for the person constructing on the PCB.

The PCB shown here has a clear layout and has been silkscreen printed to help the constructor identify the placement and sometimes orientation of components on the board. The printing is usually on the component side of the board. It also has a logo and labels informing the user of the purpose of the board.

The copper tracks can be much thinner than insulated wire links, even running between the pins of an IC where required. Tracks can also be thickened where higher currents may be expected.

The pads can be as wide as possible to allow terminals to be added, or just to make it easier for the user to solder.



The use of copper tracks instead of wires reduces the risk of short circuits associated with stray wisps on multicore wire.

Copper tracks are also less prone to vibration or breaking compared to wires. This contributes to producing a more reliable final product.

Construction

The construction manual will have a components list, telling the user what value to use for R1, R2, C1, etc.

The user can then follow step-by-step instructions on the order of fitting component and their orientation (e.g. which way to fit the IC, the diode, etc.).

This simplifies the building of the module and reduces the risk of errors.

Fault-finding

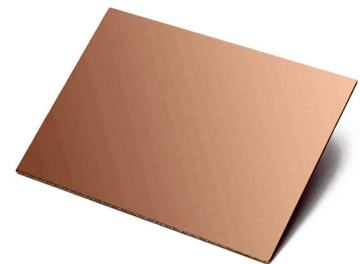
The clear uncluttered layout, combined with all components being labelled, makes fault-finding on a PCB much easier.

Creating your own PCBs

For many years, hobbyists produced their own boards

There are several methods but they all have one thing in common.

They start with a piece of copper clad board like the one shown here. You can buy them from the big suppliers like RS and Rapid Electronics in both single-sided and double-sided versions.



Using one of the methods, a copy of the tracks is laid on top of the copper.



You then immerse the board in an etching solution which eats away the exposed areas, leaving the copper tracks under the unexposed areas.

This is a messy business and uses a toxic ferric chloride solution.

The simplest method is to paint over the areas to be etched with a permanent black marker pen. This is only suitable for simple circuits and a fuller explanation is available here:

www.youtube.com/watch?v=AnOy9_1rdPw

If you have good drawing skills and are computer-phobic, this method is for you.

If you have a computer and a laser printer, you can design your own PCB layout and print it in reverse on to a toner transfer paper. The paper is placed onto the copper face with the ink side touching the copper. The paper is then ironed to fuse the toner on to the copper.

The board is then etched in ferric chloride.

Examples of this method are available here:

www.dr-lex.be/hardware/tonertransfer.html

www.electronicshub.org/how-to-make-pcb-at-home/

This one uses a page from a glossy magazine and some nail polish remover:

www.youtube.com/watch?v=cVhSCEPINpM

Another method is to print the PCB layout on to a specially formulated translucent film, such as Laserstar film. Others have reported success using an Inkjet Printer with JetStar Premium Film.

The printed film is placed over a piece of copper clad board that is coated with a photo-sensitive material. After exposure to sunlight (or a UV source if you have one), the board is etched with ferric chloride

You can buy photo resist sheets that you bond on to the copper clad board.

This shows how to use it:

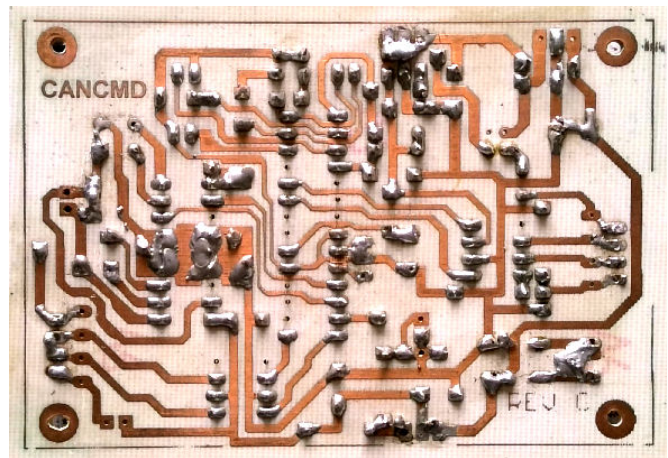
www.youtube.com/watch?v=cRCFGZxmbo0

You can also buy copper clad boards that already have the photo resist sheet bonded on to it, as this document explains:

https://static.rapidonline.com/pdf/560428o_v1.pdf

If you own a 3D printer, you may be able to convert the printing head to hold a pen and use the machine to ink out the layout on to the copper clad board.

Here is an example of a DIY etched printed circuit board, based on the CANCMD DCC Command Station sold by MERG.

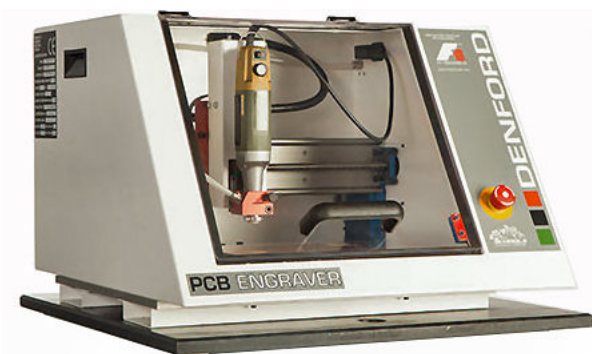


Milling PCBs

If don't want to use chemicals, you could consider using a milling machine to remove unwanted copper from the board.

You may be able to convert your 3D printer to hold a drill fitted with a tungsten carbide milling bit.

If you are lucky, you may already own a CNC machine.



With the appropriate software, and correct settings, your computer controls the milling process for you.

If you intend to make lots of one off PCBs, you could invest in a bespoke milling machine for PCBs, such as PCB Engraver shown here.

You can watch a video of it in action here:

www.youtube.com/watch?v=7pWzlk9_DYU&feature=youtu.be

Designing your own PCBs

The above methods are all still used but they are only worth pursuing if you are really pressed for time and can't wait for the turnaround time from a PCB manufacturer.

In most of methods above, you have to use a computer application to design your PCB layout. It is only a short step to go online and submit the file you create to a PCB manufacturer and place your order.

Having your design manufactured for you used to be very expensive but costs have greatly reduced, making it a preferred option for more and more hobbyists.

The PCB design file

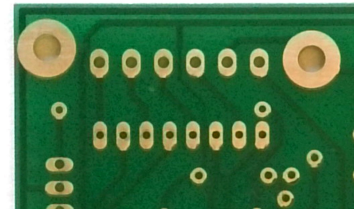
The most common file format is called a '*Gerber*'.

You can order a PCB that only has copper tracks and pads.

More commonly, you will want the manufacturer to also drill all the holes for you, specially if you are using double-sided boards that have vias.

You need a separate .GBR file for different stages of the manufacturing process.

- The copper - this is the set of tracks and pads on the board.
- The solder mask - this is a layer of lacquer that covers the whole board apart from the connection holes and pads. It protects the copper tracks from oxidation and minimises dangers of shorts during subsequent use. In this image you can just see the copper tracks under the solder mask, while the pads and holes are exposed. Many manufacturers will tin all holes and pads.
- The silkscreen - the details of all the printing that will appear on the board.



For a single-side board, you would send these three Gerber files; a double-side board would require two sets of these three. You would include the drill information for the board's holes.

Once you have created your set of Gerbers, you can produce more PCBs easily in the future, by placing a repeat order - nothing else to do.

Design software

The good news is that there are many PCB design applications out there to help you produce your Gerber files. There are also numerous tutorials and YouTube videos on the use of these packages. The MERG website also has video tutorials on some of the packages.

Note

As with all new packages, there is a learning curve to go through before you get the most from a design application.

This chapter only provides information on what is available. It would require a separate book to demonstrate all the features of PCB design software.

Free software

In alphabetical order:

Circuit Maker	DesignSpark PCB	ExpressPCB	EasyEDA
Fritzing	KiCAD	PCBweb	TinyCAD
Yenka			

Commercial software

DipTrace	Easy-PC	AutoDesk Eagle
----------	---------	----------------

Eagle CAD is available through AutoDesk Fusion 360, which is for free personal use.

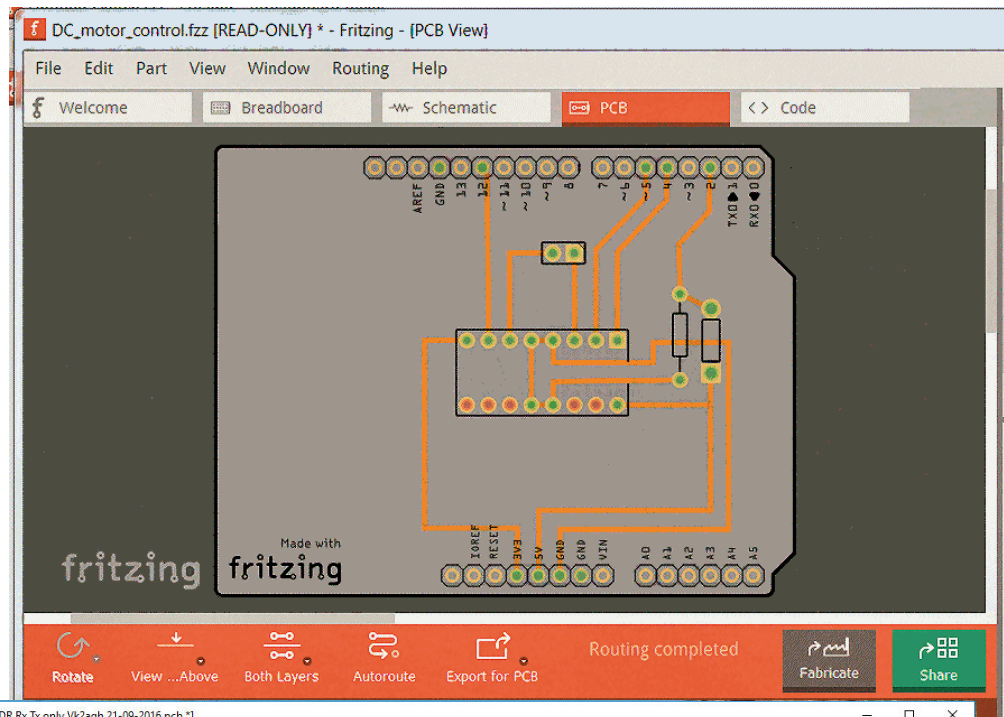
Facilities

These will vary between different packages.

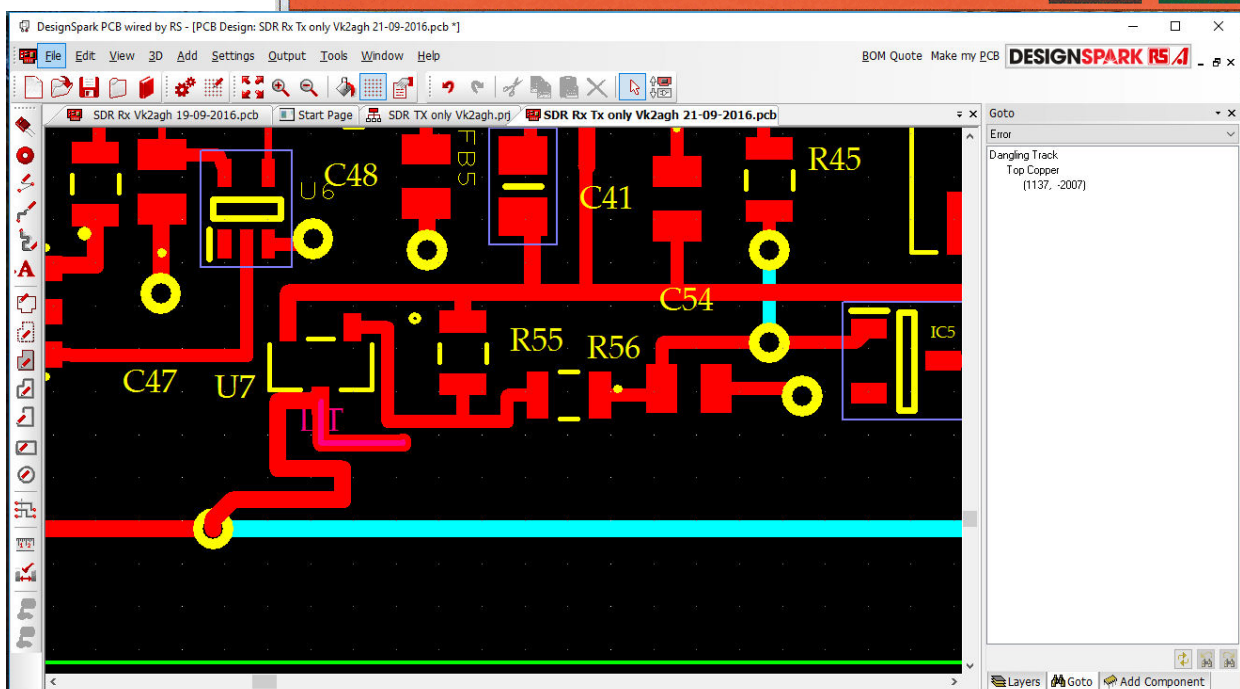
The most important ones are:

- The ability to generate a layout from circuit's schematic.
- A comprehensive parts library.
- Autorouting - the package works out the best placement for components and tracks.
- The ability to alter component placement and autoroute over again.
- Design rule checking - automatic checks on widths of tracks, spacing between tracks, the size of vias, etc.
- The ability to print out a bill of parts.
- The ability to create a 3D of the circuit.
- The extent of support / user groups / tutorials.

The Fritzing screen for the circuit used earlier with breadboards.



This is a typical screen from DesignSpark PCB.



Manufacturers

These are just a few of those willing to create PCBs in hobby-sized quantities:

www.itead.cc/open-pcb/pcb-prototyping.html

www.elecrow.com/pcb-manufacturing.html

www.seeedstudio.com/fusion_pcb.html

<https://jlcpcb.com/>

www.allpcb.com/online_pcb_quote.html

www.expresspcb.com/express-pcb-manufacturing-service/

www.pcb4u.com/

Here, for example, are some of the options on the Seeed website order form.

No. of Layers	<input checked="" type="button" value="1 layer"/>	<input type="button" value="2 layers"/>	<input type="button" value="4 layers"/>	<input type="button" value="6 layers"/>					
PCB Dimensions	<input type="text" value="100"/>	*	<input type="text" value="100"/>	* Units in mm					
PCB Quantity	<input type="text" value="10"/>								
PCB Thickness	<input type="button" value="0.60"/>	<input type="button" value="0.80"/>	<input type="button" value="1.00"/>	<input type="button" value="1.20"/>	<input checked="" type="button" value="1.60"/>	<input type="button" value="2.00"/>	<input type="button" value="2.50"/>	<input type="button" value="3.00"/>	* Units in mm
Surface Finish	<input checked="" type="button" value="HASL"/>	<input type="button" value="HASL Lead Free"/>	<input type="button" value="ENIG"/>	<input type="button" value="OSP"/>	<input type="button" value="Hard Gold"/>				
Copper Weight	<input checked="" type="button" value="1oz."/>	<input type="button" value="2oz."/>	<input type="button" value="3oz."/>						
Minimum Drill Hole Size	<input type="button" value="0.2mm"/>	<input type="button" value="0.25mm"/>	<input checked="" type="button" value="0.3mm"/>						
Trace Width / Spacing	<input type="button" value="4/4 mil"/>	<input type="button" value="5/5 mil"/>	<input checked="" type="button" value="6/6 mil"/>						
Blind or Buried Vias	<input type="button" value="Yes"/>	<input checked="" type="button" value="No"/>							
Plated Half-holes / Castellated Holes	<input type="button" value="Yes"/>	<input checked="" type="button" value="No"/>							

With planning, you may be able to squeeze two or more duplicates of your design into a standard board size, and then saw the board into multiple PCBs on arrival. Most manufacturers save you the trouble of sawing by scoring the boards so that they can easily be snapped into their separate pieces.

Some manufacturers offer a lower price if you are willing to panel share with another customer. They can optimise the use of their panels that way but you may have to wait a little longer.