

Second Year Electronic Laboratory

Electronics Construction Guidelines

Tom Clarke - Autumn 2005



“The three most dangerous things in the world are a programmer with a soldering iron, a hardware type with a program patch and a user with an idea.”

Unknown

Contents

1. Introduction	1
2. Noise Analysis	2
2.1 Supply noise	2
2.2 Supply and ground noise	2
2.3 Unwanted coupling between different parts of a circuit	3
2.4 Unwanted component characteristics	3
3. Construction methods (prototype)	3
3.1 Soldering	3
3.2 Stripboard (AKA Veroboard)	4
3.3 Ground Plane (rat's nest)	4
3.4 Verowire	5
3.4 Printed Circuit Boards (PCB)	6
3.5 Wire-wrap interconnect	8
3.6 Plugboard	8

1. Introduction

The usual reason for constructed circuits behaving in a surprising way is that they have been laid out badly. Thought given to layout before you construct can therefore pay dividends in reduced debug time.

If you follow the recommendations in this note, and have a well-designed circuit, your hardware will work.

Section 2 below gives some numbers that allow you to estimate the effect of problems due to layout at the design stage, and so determine which parts of the circuit are critical and require special care. You should read this before you attempt any layout, and again after you have completed an initial layout to check for possible problems.

Section 3 outlines a number of construction methods, giving pros and cons together with some practical hints. All are suitable for the Second Year Laboratory Design Project.

2. Noise Analysis

2.1 Supply noise

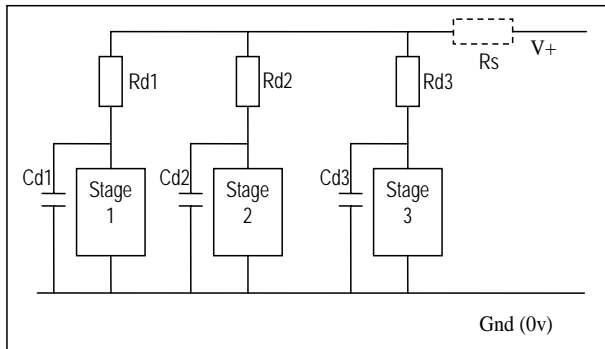


Figure 1 - RC supply decoupling

The first source of unwanted noise in circuits is a problem even with perfect layout. No power supply has zero impedance so ac current variations through one stage can lead to supply voltage variations in another. This provides a possible feedback path in amplifiers which can cause instability.

The solution is to use *RC supply decoupling* as in Figure 1, where the supply impedance is R_s . Each resistor R_d is chosen so that the supply voltage drop is acceptable - typically <300mV. C_d is then chosen so that at the signal frequency (or the potential oscillation frequency) the supply potential divider:

$$\left| \frac{R_d}{2\pi f C_d + R_d} \right| \approx \frac{R_d}{2\pi f C_d}$$

attenuates noise, and so that the impedance of C_d at the signal frequency is small (typically 10 ohms or less) and the attenuation due to the P.D. above is $\gg 1$. The advantage of this circuit is that the isolation from any one stage to another is much greater than the isolation provided by one of these filters. Typically C_d is a multilayer ceramic capacitor (say 1u) which provides low series resistance and low inductance: or for very high frequency performance a parallel combination of two capacitors.

2.2 Supply and ground noise

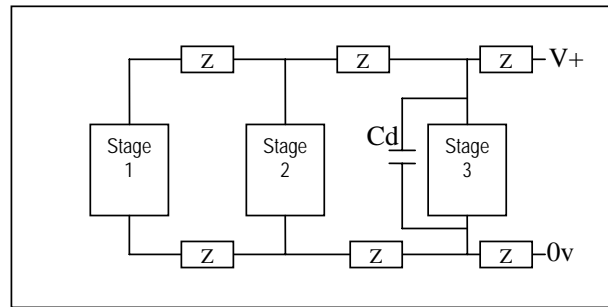


Figure 2 - supply and ground noise issues

The most common and important source of problems due to layout is this causes the circuit supplies to be noisy, and this couples noise into the circuit. Feedback paths between different parts of the circuit can easily cause unintended positive feedback (and hence oscillation) or noise from digital signal edges in sensitive analog circuits.

There are two separate issues - noise on the ground supply between different stages (which therefore will appear as extra noise on any signal passing from one stage to another) - and noise on power supplies. Both however are caused by ac circuit current. For example switching a digital output will charge or discharge the wiring and input load capacitance resulting in a current spike on either the logic supply (if charging) or the logic ground (if discharging). There are also large current spikes internal to logic ICs on clock edges. These current supplies can induce both ground noise and supply noise. Figure 2 shows the problem where the impedance of supply and ground lines (Z) will result in ac voltages at the 0v or V_+ points in the individual circuit stages if ac currents flow through the supply lines.

The solution to supply noise is simple: use RC supply decoupling to a local ground with one (or if necessary two, say 10uF tantalum and 0.1uF ceramic) capacitors, positioned as C_d in Figure 2. This provides a return path for ac supply currents in stage 3 and means that these do not cause supply and ground noise.

All complex digital ICs give guidelines on the required decoupling in their datasheets, but having more than this is useful, especially if noise problems elsewhere must be minimised. Note that the decoupling capacitor must be as close as possible to the device supply leads to minimise wiring inductance and capacitance.

A more careful solution, where very low coupling is needed to protect low-level

analogue signals, is to add a small resistor (chosen so that supply voltage drop is 0.5V or less) or inductor in series with the V+ line of each stage. This, together with a local decoupling capacitor, acts as a low pass filter to stop high frequency noise. At high frequencies the effect of ground noise should be reduced by using single-point grounding. Referenced to the common ground point each stage suffers only its own ground noise, and is not affected by other stages. These two techniques together are shown in Figure 3.

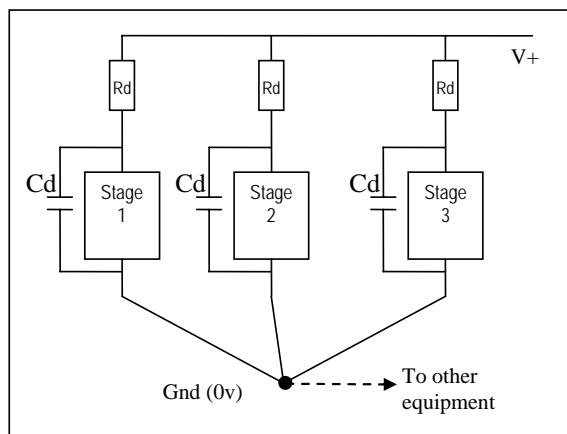


Figure 3 - single-point grounding, with decoupling

2.3 Unwanted coupling between different parts of a circuit.

Exact calculations are not usually possible, but the following rule of thumb numbers allow you to calculate order of magnitude estimates of the problems you face in real circuits. Nearly all these problems scale inversely with the distance of the two components and linearly with the length of each lead. They also reduce if components are kept close to the ground plane.

Typical "stray capacitance". For a 1cm length of 1mm diameter parallel component leads mounted 5mm above a ground plane and 1cm apart, the capacitance between leads is 0.02pF and the capacitance between lead and ground plane as 0.1pF. The lead/ground and lead/lead capacitance scale linearly with the wire diameter. The lead/lead capacitance scales inversely with lead separation, and linearly with lead length. You can work out the impedance of this at a given frequency, and therefore, for example, the a.c. current coupled into an op-amp summing junction.

For components 1cm above ground plane reduce the ground plane capacitance by 50%,

and increase the lead/lead capacitance by 30%.

Mutual (coupling) inductance of two 1cm segments of wire at a distance of 1cm is 5nH. The coupling scales linearly with wire length until length \ll distance when the coupling falls off with the product of the two segment lengths.

2.4 Unwanted component characteristics

Self-inductance of 1cm of straight wire of diameter 0.75mm is 5nH. Thus all components will have some lead inductance in series with their nominal value - this becomes significant above 10MHz, or at lower frequencies for stages with high currents.

Capacitors have internal resistance of typically 0.1 Ω or more - capacitor datasheets will provide information.

All components have some parallel capacitance due to coupling between the two leads. This is worse for radial connections than for axial.

Op-amps have limited gain-bandwidth product. All of their good properties, like rejection of common-mode inputs (equal voltage on + & - inputs), rejection of noise on supply, etc, tend to get less good with frequency. Datasheets will quantify this.

3. Construction methods (prototype)

We will consider the following methods:

- Stripboard (AKA veroboard) (0-100kHz)
- Double-sided Printed Circuit Board (PCB) (0-50MHz)
- Verewire
- Ground plane "rat's nest" (0-1GHz)
- Wire-wrap
- "Plugboard"

3.1 Soldering

Many of these methods require soldering skills, here are some guides:

<http://www.astro.umd.edu/~harris/docs/WelderSoldering.pdf> (technical & detailed)

<http://www.epemag.wimborne.co.uk/solderfaq.htm> (good guide for novices)

3.2 Stripboard (AKA Veroboard)



Figure 4 - stripboard

This is the method beloved of hobbyists because it produces a reasonably neat and robust result for minimum cost. There are some problems when stripboard (see Figure 4 above) is used for prototyping and circuit development by inexperienced users.

- The strips of copper can become detached after several changes of component.
- It is easy when breaking copper strips to leave an invisible hairline of copper that results in continued (possibly intermittent) connection. A special hand tool is available to break strips at any hole, or a drill-bit can be used.
- The relationship between components and the circuit diagram is often obscure - since the connecting strips, and where they are broken, cannot be seen from the component side. This means that constant cross-reference between the layout diagram and the circuit diagram is required when debugging or testing.

The most serious problem is that this method tends to give very poor high-frequency performance. High impedance ground connections (when a single strip, or several strips in series, convey 0v to different stages of the circuit) result in circuit ground noise which cannot be reduced through decoupling. Adjacent strips have a high capacitance (typically which can result in unwanted

coupling. These problems will affect even low frequency designs which use components with high gain above 50kHz.

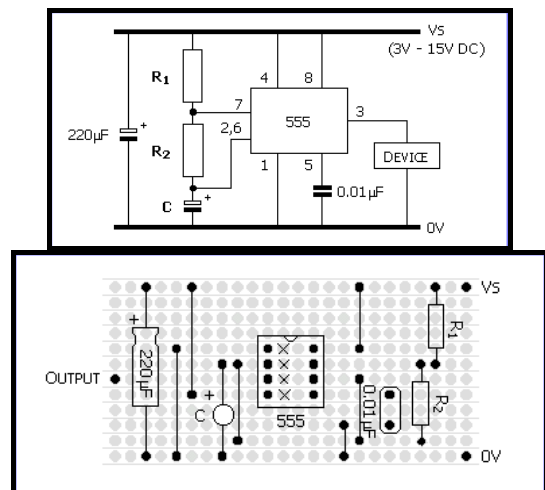


Figure 5 - stripboard layout

As illustrated in Figure 5, stripboard layouts have the strips running parallel with the supplies, and usually look a little like the circuit diagram, with separate stages laid out horizontally. However this can lead to a long thin layout, in which case two lines of components, each with its own supplies, can be doubled up.

For good performance make sure:

- Signal connections run on strips for the minimum possible length. Always break strips carrying signals as soon as possible to reduce the radiation and pickup.
- Do not position strips with inputs and outputs next to each other.
- Use multiple strips in parallel for ground connections to minimise impedance. In fact it is good policy, if hf performance is important, for all unused strips to be wired together and connected to ground - so making a sort of distributed ground net that approximated to a ground plane.
- If necessary protect inputs from outputs with guard strips in between which are connected to ground.

3.3 Ground Plane (rat's nest)

Ground plane layout has many advantages when prototyping, although you need good soldering skills and must use a neat layout with short leads or the circuit can degenerate into a mess which is not robust.

The big advantage is that with a complete copper ground plane beneath the circuit high frequency performance is excellent, ground noise is effectively non-existent and with good supply decoupling the designer can stop worrying about the most common sources of noise. In fact performance is much better than even two-sided PCB. Good construction in three dimensions will minimise lead lengths and therefore pickup and radiation of noise.

Ground plane construction is also excellent for debug and test. All of the circuit is accessible to probes, and the relationship between the components and the circuit diagram is clear because all connections are visible. Circuits can be changed and adapted much more easily than with the other methods: with the proviso that neat and careful layout is needed or the circuit quickly degenerates into a spider's web. Analogue and digital ICs can be mixed, although large numbers (more than 100) of digital connections can be problematic, and are better routed either using a PCB or a digital technique such as wire-wrap.

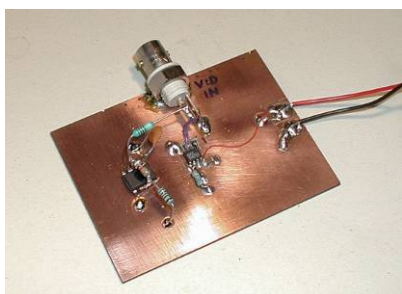


Figure 6 - ground plane circuits

Figure 6 shows some examples of good ground-plane layout. Circuits should be built on ground plane and laid out neatly like a circuit diagram. In some cases, due to component sizes and circuit topology, a 3-dimensional construction may be neatest. All components should have at least one lead securely anchored to ground or another component which has short leads and therefore is well anchored. It is vital (see below) that leads of components which connect to signal nodes should be short.

Leads connected to supplies, ground, or decoupled voltages are less critical, but should still be reasonably short to make the layout stiffer and more robust. Well-constructed circuits will survive being dropped on the floor, having scope probes connected at any point, etc, without deforming.

Some useful tips for ground-plane construction are:

- DIL Integrated circuit must be anchored at opposite corners if possible, usually the two supplies will enable this. Use small decoupling capacitors (0.1u multi-layer ceramic) with short leads to anchor supplies not connected to ground. These help to provide good a.c. supply decoupling as well as providing mechanical rigidity. Pins other than the supplies can be bent horizontal to the package for easier soldering.
- Digital connections can be made using thin enamelled wire, hooked round the integrated circuit leads. The correct enamel (usually pink or red in colour) burns off when the wire is soldered.
- Inputs & outputs can be made via connectors anchored securely to the ground plane - BNC connectors are particularly useful for single signals.
- A "front panel" can be made from another ground plane board a right angles to the base with small squares of board on either end providing rigidity, the whole soldered together
- Make sure you have the correct soldering iron. This is needed to solder to the ground plane itself which acts as a heat-sink. The iron should have a reasonably large bit, be at least 60W, and have temperature control.

3.4 Verowire

This uses very fine insulated wire which is threaded through plastic 0.1" pitch press-on combs using a wiring pen and wrapped round the adjacent terminals of ICs. The pen, wire & combs are available from RS. The method has poor HF characteristics, but works well for a small number of digital signal connections. Care is needed if supply and ground noise in digital circuits is to be

minimised. Figure 7 shows how Verowire works.

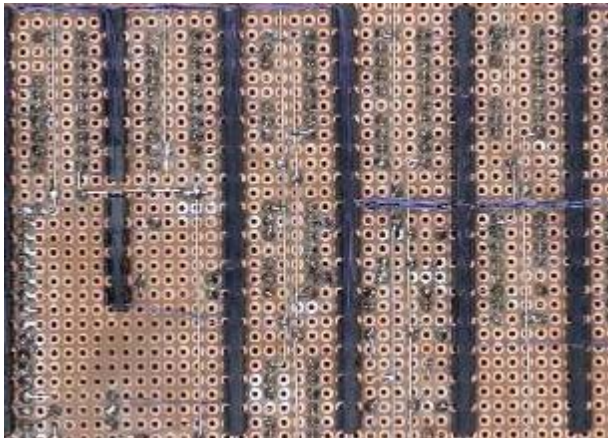


Figure 7 - Verowire circuit

3.4 Printed Circuit Boards (PCB)

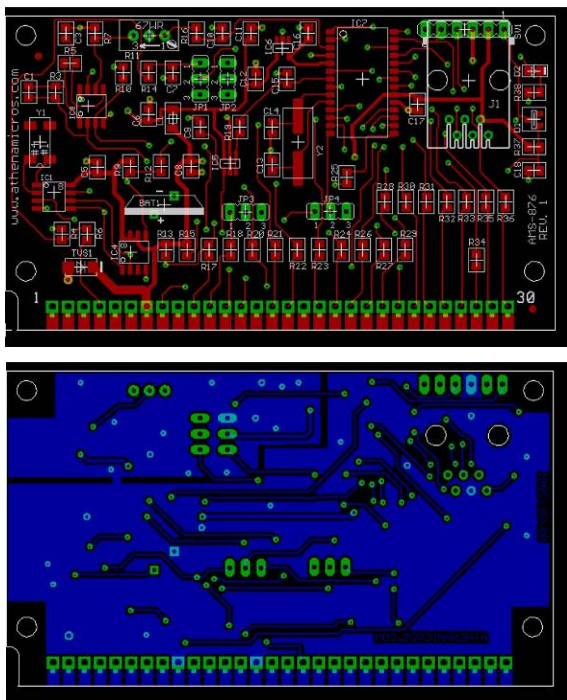


Figure 8 - Two-sided PCB layout with silk-screen

For high-performance fast digital circuits, with many thousands of pins, multi-layer PCBs (at least 4 layers) are the preferred choice. This allows two layers for routing with supply and ground connections each having a whole layer. The result is excellent noise characteristics and relatively easy and compact routing, which is usually done automatically.

Analog circuits are usually implemented with good performance on two layers, even if there are also digital ICs, providing the total

routing requirements are small. In fact simple circuits may only require a much cheaper one-layer PCB. Figure 8 shows a typical use of 2 sides. One of the sides is used for short tracks where the routing side needs them, but mostly has a ground plane providing good high-frequency characteristics. Sophisticated CAD software will insert such a ground plane automatically.

Where a two-layer or more PCB is used *plated-through holes* (PTH) are strongly recommended. These allow vias (connections between sides) to be manufactured with the board, allow tracks to connect to pins from both sides of the board, and just as importantly the plating through each hole means that the copper on either side of a component pin is reinforced and much less likely to delaminate. Without this plating, component pads must be much larger to stop the copper from lifting during soldering.

Silk screening is used to print (usually the component side) with component IDs etc which help correct placement of components. It is generated automatically by CAD software along with the signal tracks, and can be annotated as required.

PCB layouts can be made using CAD software. One recommended option for simple circuits is [Eagle CAD](#). The Eagle-light package is sufficient for most purposes and can be downloaded free for non-commercial use.

The department has limited in-house facilities for PCB manufacture, one or two sided, no PTH, minimum track (or gap) width 0.04 mm. Holes must be drilled by hand. Eagle output printed on transparency sheets can be used by the technicians. For more information contact [Neil Todd](#).

Alternatively the manufacture can be outsourced. Typical specification is: two-sided, PTH, drilled any number of sizes & holes, solder mask, silk screen legend on one or two sides, minimum track or gap 0.2mm.

A number of suppliers will provide 1-off small two-layer prototype boards with PTH for a competitive price, or much quicker if more is paid. Table 2 gives some examples. The size specified is the minimum price, larger sizes will cost more. Note that in some cases multiple boards can be specified as a single job to reduce costs: see web sites for more details.

Note also that faster turn-round is possible but more expensive.

Company	Specification: all 0.02mm track & gap min spacing or better	Speed (wk. days) incl delivery	Size. dm =100mm	1 off cost	Delivery	Total	Notes
www.pcb-pool.com	Two-sided, PTH, no mask, no legend	9	1dm ²	£19	£6	£25 + VAT	+£10 mask +£10 1Xlegend
www.olimex.com	Two-sided PTH, maskX2, legendX1	8-19 (typ. 10)	100X160mm	£19	£4	£23	From Bulgaria - will pay VAT on delivery. Payment via CC only

Table 2

3.5 Wire-wrap interconnect

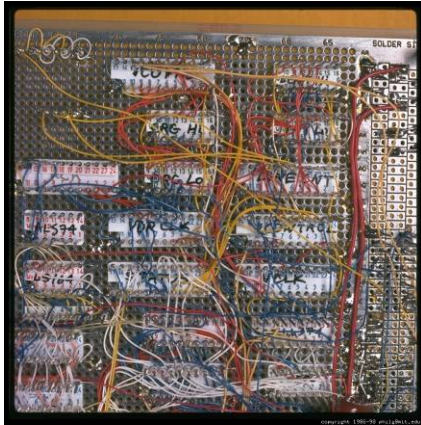


Figure 7 - connection side of wire-wrap board

Only suitable for digital circuits this technique requires special wire-wrap sockets for all connected components, these are normally mounted on perforated board with an integral "colander" ground plane which is separated from the holes. All connections are made via thin insulated wire-wrap wire, with a special tool. The merit is that complex interconnect can be made quickly and changes are possible at any time without making the circuit less robust. But the cost and inconvenience of wire-wrap sockets, together with the difficulty in adding analog components, make this method less attractive.

Tips:

- Put analog components onto DIL headers mounted on a w-w socket
- Ensure good ground by using prototype board with colander ground and inserting w-w pins as necessary to make gnd connections
- Use electric wire-wrap tool for faster and cleaner wrapping

3.6 Plugboard

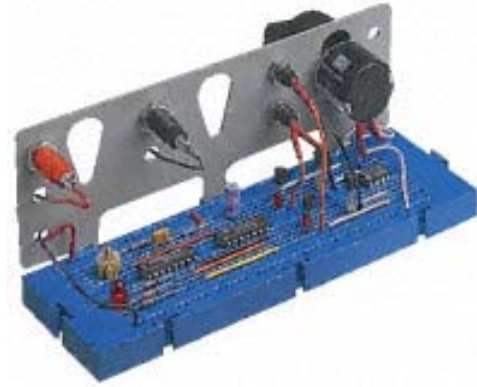


Figure 8 - "professional prototyping board" from RS

Good for very quick low frequency prototyping, circuits constructed in this way are not very robust because wires can easily fall out, or be pulled out. But it is fast and changes can easily be made. Not suitable for high density digital designs with more than a few hundred interconnects because the relatively thick connection wires become unwieldy.

Analog circuits will work, with the same layout issues as for stripboard, but generally high frequency performance is less good.