

4 Designing Printed Circuit Boards

4.1 Introduction

Once the circuit diagram is available for a PCB, then the process of designing or '**Laying Out**' the PCB can commence. In the past most layouts were produced manually by applying fine black adhesive tape and pre-printed sticky pads to a clear film or mylar. The resulting artwork is photo-graphically reduced for use in PCB manufacture. Today most PCBs are produced by CAD packages which produce artwork on a plotter or printer. We will only be examining techniques applicable to layout via CAD tools.

Usually a netlist generated from the schematic entry package forms the starting point for the layout. However a PCB layout can also proceed manually from a circuit diagram on paper. The netlist saves time and avoids human error in the process of manually entering of components and connections.

4.2 Placement of Components

It is extremely important to place components in such a way as to minimise the length of interconnection or track. The longer the tracks the more PCB real estate is used and the larger the PCB. If two components have a large number of intercomponent connections, then they should be placed next to each other. The same applies for groups of components.

4.2.1 Manual Placement

When manually placing components using a CAD package it is easy to see where the interconnects between ICs are the greatest. Figure 1 shows a typical display of components with the 'unrouted' connections shown. This display is commonly called a **rats nest** for obvious reasons. By moving around the components the 'rats nest' can be minimised corresponding to short interconnections between adjacent components.

Components are usually oriented orthogonally on the PCB. For dual in-line integrated circuits they are usually oriented in one direction with pin 1 toward one side of the board. Where they are placed horizontally or vertically, all ICs in a particular direction should have pin 1 facing the same direction. The reason for this is that automatic component loading machines have limited rotational abilities. Also testing and debugging is easier if all ICs are oriented in the same way.

4.2.2 Automatic Placement

It is usually easy for a human to minimise the rats nest display, especially if he/she has some knowledge of the circuit structure. Also sometimes there are mechanical and electrical constraints to consider.

A different approach is to let the CAD package optimise the placement. This is done in a similar way to the manual process. The length of interconnects between components is calculated and minimised. Where the process differs is that usually the automated process works to a grid. The user specifies a grid spacing on which the components are positioned. Usually the program does not consider the

component size when choosing a position. Often two ICs can be placed in the space used by a large IC.

A combination of automatic and manual placement can be used. Some components can be fixed manually. Then a grid can be specified and automatic placement can proceed. After this the placement can be further refined manually.

4.3 Power Supply Routing

It is usual that good connection be made to the power pins of components. Long thin tracks are inductive and resistive in nature. This results in a low and varying supply voltage that can cause components to malfunction. Decoupling capacitors placed close to the component can minimise these unwanted effects and need to be added to the circuit if not already shown on the circuit diagram. Usually the thickness of power tracks is made larger than those used for signal tracks to further reduce these effects.

For a double sided PCB, tracks on the component side are at right angles to those on the solder side. Power supply tracks must also obey this rule. The best way to obtain a good supply to all ICs is to use a matrix of power lines as shown in figure 2. Main feed lines run parallel along the edge of the PCB on the solder side. Parallel lines run off these feeds at right angles on the component side and pick up the supply pins of the ICs. Most dual in-line (DIL) digital ICs have corner supply pins and the supplies can be used in pairs in between the two rows of pins. Decoupling capacitors can be placed at each IC across these bused lines.

On multi-layer boards entire planes are allocated to the supplies and routing is not required. The CAD package makes automatic connection to the plane by placing pads on the appropriate layer. On a single PCB good power supplies can be difficult to achieve and links may be required. Often bus-bars as shown in figure 3 can be used.

4.4 Routing of Tracks

4.4.1 Track Size

The standard spacing of pins is a multiple of 1/10 of an inch. Normally only one track is run between IC pins spaced at 0.1 inch. For a track width of 12 thou (0.012") and spacings either side of the track of 12 thou this leaves about 60 thou for the pad around each pin. It is possible to run two tracks between pins, however the track width must be made less than 10 thou. This should be avoided as the PCB manufacturer often charges 'danger money' to produce such PCBs.

4.4.2 Manual Routing

This is merely the selecting of one connection point (with a pointing device such as a mouse) and moving to another while clicking on the corners in the track. When the layer is changed, a via is normally inserted automatically. The orientation of the tracks is normally horizontal, vertical, or at 45° (mitred).

4.4.3 Mitred Tracks

When routing buses (data, address etc), mitred tracks are used as shown in figure 4. The purpose of the mitre is to allow the tracks to move around pins without obscuring the path of an adjacent track. This technique is primarily used for routine memory arrays, where a whole set of connections must run through a set of ICs.

It is also possible to mitre right angled corners for ascetic reasons, however it is not normally required.

4.4.4 Auto-Routing

Modern CAD tools usually include some form of auto-routing of tracks. The sophistication varies between packages. It is often difficult for the program to realise that it must leave a path clear for a different connection to get through. Some basic routers cannot automatically move a track once it has been routed. Other incorporate a 'rip-up router' which enables some modification of the existing tracks to get a track through.

Several stages exist in the process of auto-routing.

Reordering of Nets

- Nets re-ordered so that closest connections are adjacent after placement.

Power routing

- Nets connected to supplies are routed

Memory routing

- Used mitred corners to route buses

General routing

- Route the rest of the connections

Minimise vias

- Where a via is close to a pin attempt to remove it by swapping track to other layer.

Note that often auto-routing an entire PCB can result in only a portion of the tracks routed. The remaining connections need to be manually routed. Usually this becomes impossible as the auto-router has used up possible paths and wire links or additional layers are required. It is often better to auto-route small sections of the PCB, and check that the buses, in particular, have been routed in an efficient manner. One of the most important things to do is to save the work before each stage of auto-routing. This allows you to go back to what you had before tracks were incorrectly placed.

4.5 Checking of PCB

Most CAD packages can check the routed PCB with the original netlist and report on any differences. These can be rectified in the layout or perhaps the schematic needs to be changed as with additional bypass capacitors. Another source of difference can arise if identical gates in an IC have been swapped to produce an easier PCB layout. The pin numbers of the schematic need to be changed. Some packages do this automatically and this is called **back annotation**.

Further dimensional checks known as **design rule checks** can also be made. This checks if minimum spacing constraints have not been violated. Such a violation may result as a short on the finished PCB. Also the minimum track width is important as this can result in open-non-connected tracks.

4.6 Artwork Generation

All CAD packages have some form of output. The standard output is known as a **gerber** output. A file is produced which can be sent to a gerber plotter. This plotter draws with light onto photosensitive film instead of using ink onto paper. It has a number of apertures or circles to use which flash a disk of light onto the film. These disks form pad and vias. Tracks are drawn by moving a (12 thou.) aperature across the film. When finished the film is developed to show the track and pad outlines.

All the layers required to manufacture the PCB (section 3.3) are generated in this manner. A drill file known as an **excellon drill tape** is also generated by the CAD package.

With the increasing resolution of laser printers, output can often be obtained on these devices. Output on dot matrix printers is usually only used for visually checking a PCB.

Revised:

C R Stephens 1996, 2002 August 21

G. Tobin 2007 July 20

*CRS:men 12 August, 2007
Designing-PCBs.odt*

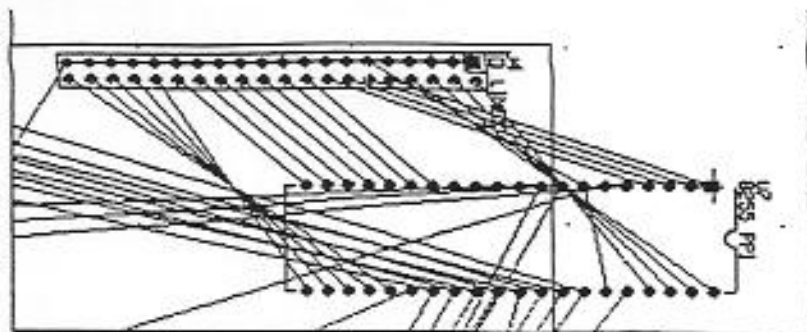


Figure 1 Rats nest of connections

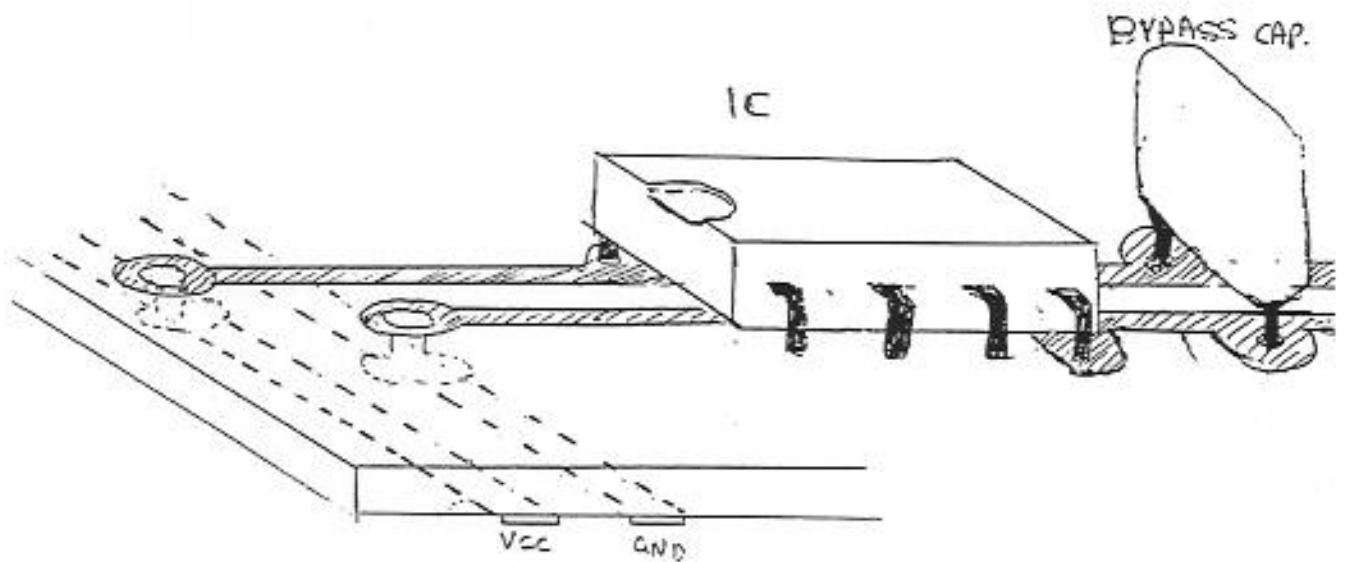


Figure 2 Power connections

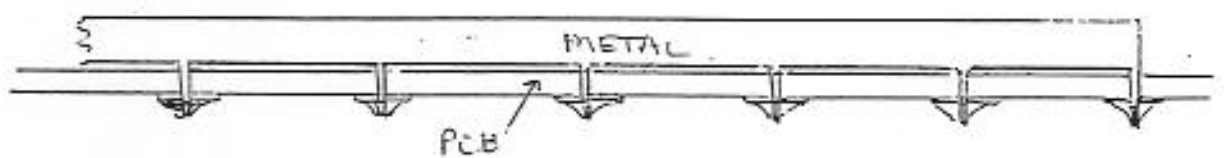


Figure 3 Buss-bar

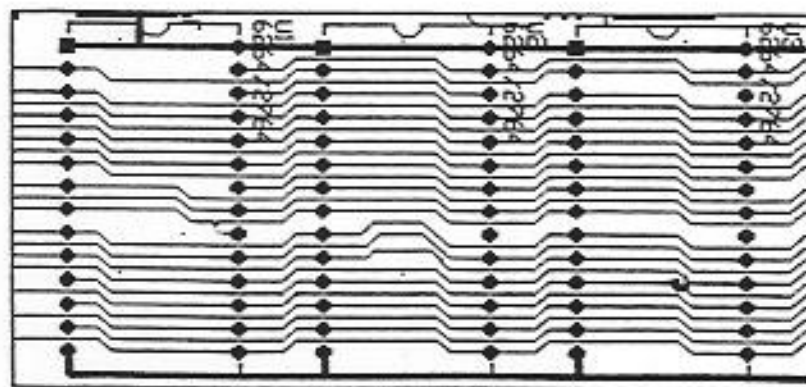


Figure 4 Mitred tracks for memory