ELE32CMP/ESP/OPP/BMP

PCB Design

Introduction

Almost all commercially released electronic products include a Printed Circuit Board (PCB). A PCB is a means of mounting and connecting most but not all the components that comprise an electronic product. A PCB can also be used as a prototyping tool and as such may be suitable for many student projects this year. Wire wrap is the other main prototyping technique available. Wire wrap may be a better choice for digital prototyping than PCB, especially for complex digital circuits (e.g. 32 bit microcomputers and DSPs). Some of the advantages of wire wrap include:

- Direct point to point connections which result in fewer parallel wires and thus less coupling.
- No need to perform a layout with a CAD package. However careful thought is still needed to achieve optimal component positioning which will give shorter and less overlapped wiring.
- Modifications are possible if some forethought is given to the order in which wraps are applied to a given post.

Wire wrap is not suitable for many circuits, especially those which are sensitive to small voltage or current changes. This is due to the highly inductive nature of wire wrap wire (thin) and the lack of grounding and shielding.

There are many types of substrate from which a PCB can be made including:

- Fiberglass is the normal and cheapest PCB substrate material. Its main disadvantage is its high dielectric loss which becomes important at microwave frequencies (greater than approximately 1GHz).
- Teflon has a low dielectric loss and is used at microwave frequencies. Another advantage of Teflon is its high electrical resistance even in moist conditions as it is not able to absorb water, thus making it suitable for sensitive analogue instrumentation. The main disadvantages of Teflon are its poor mechanical properties (strength, rigidity and creep) and price.
- Alumina (aluminium oxide) is used almost exclusively in microwave circuits due to its low dielectric loss. Alumina can be used at high temperatures however it is brittle and expensive. It is difficult to manufacture large alumina PCBs.

On each of these substrates copper is the normal track material. This track is often coated with tin, palladium, silver or gold to improve corrosion resistance and ease of soldering. At high frequencies current is conducted primarily through the outer layer of the track (`skin effect') and thus it is important to keep the surface free of corrosion.

One and two layer PCBs can readily be made both commercially and at La Trobe. For all but the most complex digital boards two layers are usually sufficient. Complex multi layer boards can be commercially made (with over 20 layers). These complex boards may even contain some components inside the board itself. For thermal or mechanical reasons a solid copper inner layer may be used.

When a PCB is to be made commercially there are two types of costs involved:

- Quantity independent costs. These include artwork preparation and tooling. The artwork costs can sometimes be reduced by doing some of it yourself. The tooling cost depends on the complexity and size of the PCB including the number of holes, number of different size holes and the number of layers. This cost may range from several hundred to several thousand dollars.
- Quantity dependent costs. This is the cost per board which normally depends on the size and complexity of the board. This cost may also drop when ordering large quantities. This cost normally ranges from ten dollars upwards.

It is the quantity independent costs which normally prevent 'one off' prototypes from being commercially made and is also the reason all student project PCBs are done at La Trobe.

Independent of where the PCB is to be made a CAD package such as Protel is required to layout (draft) the artwork which is photographically transferred to the PCB before the etching step. Protel allows a PCB to be designed from a schematic via a net list. Various levels of automation are available including automatic component placement and automatic routing. However a large amount of manual intervention is usually required. Some sections of a PCB may not be laid out from a schematic such as a microwave matching network. A PCB can not be easily modified thus time and effort should be spent to ensure the same connectivity on the PCB as the original schematic. The schematic should be correct in the first place since the circuit should have been simulated. During prototyping it may be easier to construct several smaller PCBs than one large one. If there is a design problem only a small PCB needs to be remade. However a final design should consist of as few PCBs as possible as this reduces interconnects and overall costs.

Component placement

If a PCB is required to be a particular shape or size then these dimensions should be used to draw a keep out layer. If mounting holes are required then they should be placed correctly and the required clearance also shown. If no dimensions are given then rough calculations should be done to determine a suitable size rectangular area. Some PCB manufactures use a set of standard sizes which should be used to minimize cost. The number, size and position of mounting holes used will depend on the PCB size and weight of the components to be placed on it.

After the physical dimensions of the PCB have been determined the components to be mounted should be placed starting with the large or critical ones and finally the small ones such as resistors. Generally better performance will be obtained if the analogue and digital areas (if a circuit has both) are separated as much as possible. Analogue inputs and outputs should also be spaced as far apart as possible to improve stability. Decoupling capacitors are more effective (especially at high frequencies) if they are placed close to the component they are decoupling (reduced series inductance and resistance).

Some heavy bulky components would be best mounted off the PCB such as power transformers. In the case of mains components this would also reduce the amount of 50Hz coupled into the rest of the circuit.

Track thickness and separation

For low frequency circuits the characteristic impedance of tracks, hence their width, is not important. A thicker the track is generally better as this reduces series inductance and resistance. Thicker tracks are also easier to manufacture with cruder production techniques. However thicker tracks means less separation between tracks for a given number of tracks per area of board. As track separation is reduced inter track capacitance and coupling increases. This is an important consideration for higher frequency circuits and digital circuits with fast logic transitions. As a general guide, start with 30mil analogue tracks and 12mil digital tracks. If a track is to pass between normal 100mil spaced pins then it should be tapered to 10mil to allow sufficient clearance. If high currents are involved, such as in the output stage of a power amplifier, tracks of several 100mil should be used to reduce voltage drop and heating.

At UHF and microwave frequencies the track width is critical as this determines its characteristic impedance and thus how energy is propagated. Track thickness and lengths should have been calculated using the PCB's characteristics (ε_r and thickness) with a suitable simulation program such as Touchstone. The results of this simulation can then be used directly in the PCB layout. For applications where track thickness is critical the tolerance of the manufacturing process should be known as this will determine the thinnest usable track.

Tracks should normally be placed vertically, horizontally and at 45^0. This simplifies the final layout and allows for high track densities (number of tracks per unit area).

Power and ground tracks should normally be placed first followed by critical tracks such as digital clocks and feed back paths. The remaining tracks can then be placed such as to minimize their length.

Pads, holes and vias

Protel provides footprints for most of the standard components. A footprint consists of a components boarder representing its physical size and a collection of pads. For leaded components each pad will have a hole in its center. This hole should allow a firm fit for the leg to be inserted into it. If the hole is too large excess solder will be required complete the joint, thus reducing its reliability. The default hole size for many components may not be large enough and should be checked. If the hole size is increased the corresponding pad size may need to be increased. If the hole in a pad is made too large for the pad insufficient copper will be left for a reliable solder joint. If the PCB is made at La Trobe this may also result in the pad being removed during the drilling stage. If a board is to be commercially made as few hole sizes as possible should be used as this significantly effects the tooling costs.

Vias are used to connect one layer of a PCB to another. For a two layer PCB a via passes from the top to the bottom layer. For multilayer boards a via may pass through all layers or only some and thus may not be visible on the final product. Since a via involves drilling it adds costs to a commercial PCB. Commercial vias are plated through hole (the connection between layers is provided by a thin layer of copper deposited after the drilling step). For vias in PCBs at La Trobe wires are soldered between the top and bottom layers. Therefore via placement differs between commercial and La Trobe boards. Plated through hole vias can be placed anywhere including under components and component pads where as soldered vias can only be placed in areas accessible by a soldering iron. Where possible, legs of leaded components should be used as a connection between layers.

If a component does not have a footprint in the Protel library it must be created and saved, either in the main library or a separate one. This footprint should be accurate and include a boarder slightly larger than the actual components mounted surface area. Mounting holes (if any) should also be included.

Power and ground planes

The use of power and ground planes can simplify the layout of a PCB and provide a lower impedance power source to all components. The lower impedance is a result of a decreased series inductance and resistance and an increased parallel capacitance due to the use of an entire plane compared to individual tracks. This lower impedance will offer dramatically improved high frequency performance which is important for modern complex microprocessor based circuits which may use clock speeds greater than 50MHz. The use of a power and ground plane requires a board to have at least four layers and therefore can no longer be manufactured at La Trobe.

If only two layers can be used then a ground but not power plane should be used. Such a ground plane can be either the top or bottom layer with the other layer being used for the signal tracks. The power is then provided by thick tracks on the signal track layer. Such power tracks should be placed before any signal tracks and made as thick and short as possible. The ground plane may need to be broken by either signal or power tracks if the circuit is complex. If the ground plane is to remain effective all sections should be connected by low impedance (wide) tracks. If the ground plane is to form part of a microstrip circuit it must not be broken and should extend past all microstrip lines and components by at least one wavelength. If the complexity of a digital circuit is such that a ground plane is not feasible then wide (at least 30mil) tracks should be used for ground and power distribution. Ground and power should be placed on opposite sides of the PCB and run parallel thus reducing the effective series inductance of the power tracks.

Thermal considerations

The power dissipation of components should be calculated or simulated so that their maximum power ratings are not exceeded. The maximum power rating of most components decreases as the component becomes hotter. How hot a component becomes depends on how much heat it is required to dissipate (the result of the above calculation or simulation) and how effectively the component can dissipate heat. The efficiency of heat dissipation is dependent on the component and/or its heat-sink mounting. Consider a 1W power resistor mounted flush against a PCB and connected via thin tracks. If such a resistor were required to dissipate 1W it would overheat and become brown. The life expectancy of this resistor would decrease and the solder joints connecting it to the PCB may become brittle as the tin portion of the solder evaporates thus causing intermittent or faulty behavior. If the same resistor were mounted above the PCB (say 10mm) its body would become cooler by increased convection and its legs would add to the cooling effect and their thermal resistance would leave the solder joints cooler and thus more reliable. The solder joints could be further cooled by increasing the width of the connecting tracks.

The amount of power a transistor can dissipate before a heat-sink is needed depends on its case. A small plastic case (e.g. TO-92A) could dissipate 300mW and a large metal case (e.g. TO-3) could dissipate 1W before a heat-sink is used if the ambient temperature was less than 30^0C and good ventilation around the PCB was provided. If a heat-sink is used increased cooling is provided by having it vertically rather than horizontally mounted. Increased cooling can be obtained by using forced air cooling (i.e. a fan) such as in a computer power supply or on a CPU. This many not always be the best option as a fan is a mechanical device and may not be reliable enough. A transistor should have its power correctly de-rated or its life expectancy will be markedly reduced. For most power transistors manufactures provide such data.

A PCB can be miniaturized by replacing some or all its leaded components with their equivalent surface mounted ones. However these components will normally have reduced power ratings and will get hotter for the same power dissipation. Power surface mount components will often require a large power or ground track to act as a heat-sink.

For high reliability designs the thermal expansion of a component's case should be matched to the thermal expansion of the PCB. This is the reason some ICs are supplied in ceramic packages. For PCBs requiring large power dissipation a solid copper inner layer can be used such as with the CRAY-YMP.

Mechanical considerations

Many components come in several packages or a given package can be mounted in different ways. Many electrolytic capacitors come in both an axial and radial form. The axial form will occupy more PCB real estate but if soldered flush will offer increased mechanical stability. For components with large bodies and thin legs such as crystals they should be mounted horizontally and a supporting wire soldered to their case. If a power transistor is mounted vertically and attached to a heat-sink (to gain the maximum cooling effect) the heat-sink should be firmly attacked to the PCB and not supported by the transistor. Often such heat-sinks are mounted horizontally and screwed to the PCB directly (with a reduction in cooling effect but increased mechanical stability).

IC sockets are often used for components that may need replacing or programming. The type of socket will depend on how often the inserted device needs to be replaced. Turned pin sockets will offer a better connection than spring brass but at increased cost. For PGA devices (such as most 32 and 64 bit processors) a Zero Insertion Force (ZIF) socket should be used to reduce the chance of possible mechanical damage. However all sockets will increase the inter pin stray capacitance and increase the series inductance and resistance of each leg. This is usually not a problem except in very high speed circuits such as those using ECL. Some sockets have built in decoupling capacitors which can offer improved performance, at a price.

As mentioned in the introduction mounting holes should be one of the first things placed on the PCB. Such holes should be placed with sufficient room for screw heads, washes and the screwdriver head. They should normally be placed at the edges of a board unless it is sufficiently large (i.e. a PC mother board) where some middle support may be required. The position of heavy components, such as power transformers, should be carefully thought out. Support holes should be included as close to heavy components as possible. If such components are to be mounted on a vertical PCB their solder connections alone should not be used as support.

Easy access to screwdriver adjustable resistors, capacitors and inductors is important. Many adjustable capacitors and inductors have vertically accessible adjustments which allows them to be positioned anywhere on the PCB. However some adjustable resistors have their adjustments such that horizontal access is required. These components should then be mounted at the edge of the PCB.

Copper balancing of a PCB is important in high quality commercial products. This involves maintaining an even distribution of copper area throughout a layer and between layers. The use of power and ground planes can help reduce the effects of a copper imbalance. If a large ground plane is place in one area of a PCB and sparse tracks in another the PCB may warp with time and temperature variation due to the differences in surface tension between these areas.

Connectors and cabling

Wires should not be directly soldered onto a PCB as they are easy broken off if moved. This is especially true for thin wires. Thicker wires may not break as easily but the pad to which they are soldered may peal off the PCB. Attaching wires directly to a PCB also makes it harder to assemble, repair or test.

Connectors should be used for all power and signal wires connected to a PCB. The choice of connector depends on the type of power or signal involved. For signals that require a specific source or load impedance the appropriate impedance connector and cable should be used such as a 50Ω BNC connector and RG58 coaxial cable for thin Ethernet signals (see the Ethernet card on a PC). This is particularly important for high frequencies where reflections become more important. For parallel digital signals such as a SCSI bus IDC connectors and flat ribbon cable are often used. By earthing alternate wires in such a cable coupling between signals is reduced. Many signals have standards for connectors and cables such as RS232 for a serial interface. Unless there is a good reason not to these standards should be adhered to. There are many possible choices for power connectors. A connector should be chosen to meet the peak voltage and current requirements of the circuit. Where possible connectors should have a unique way in which they can be joined. This is often achieved with a small indentation or other such key. On a PCB each connector should be different thus avoiding incorrect wiring. Sometimes the other sex of the same connector can be used to increase the uniqueness. If this is not possible the consequences of making a wiring mistake should be analyzed and possible harm reduced by careful choice of the pin out of each of the connectors.

In some cases, especially on small PCBs with large connectors, the connectors can be used either by them selves or in conjunction with mounting screws as the mechanical support. If this is the case such a connector should not rely solely on the solder joints of the signal pins for support. Many connectors are provided with screw attachments or additional heavy-duty solder pins for this purpose. If connectors are used to support a PCB and the PCB is to be mounted in a case then their position on the PCB is critical and they should be placed at the same time the board dimensions and/or support holes are drafted.

Lights and switches

Some circuits require a combination of switches, buttons and lights. If these devices are to be available on the front panel then they could be either mounted at the edge of the main PCB in a right angled fashion or a separate front panel PCB could be constructed. If mounting on the main PCB is chosen the design would be limited to one or at most two layers of front panel control. Tight restrictions would also be placed on the vertical position of the main PCB. However, if a separate PCB is to be used extra connectors and cabling would be required. The extra PCB may also add considerable cost to a commercial product due to the extra tooling costs. Some switches can be used for PCB support provided the same precautions are followed as for the connectors (see Connectors and Cabling).

Some lights and switches will not require PCB mounting, such as power switches or large panel mounted lights. The panel to which these devices are attached provides all their mechanical support and thick wires should be directly soldered to them and terminated with an appropriate connector.

PCB layout checklist

The following is a simple but not comprehensive check list to layout a PCB.

1. Establish where the PCB is to be made and thus if any special restrictions apply.
2. Determine the size and number of layers of the PCB required.
3. Place all mounting holes and other components for support.
4. Place alignment markers on each layer.
5. Place the large, heavy and critical components first.
6. Place all other components. Remember to separate analogue and digital areas.
7. If there are any shape critical tracks such as a microstrip lines these and their corresponding ground planes should be placed first.
8. If power and signal planes are not used then their corresponding tracks should be placed.
9. All other tracks should be placed in order of importance.
10. Check the layout corresponds exactly with the schematic.
11. Check the layout to ensure that all components have sufficient physical clearance.
12. Check that all screwdriver adjustments can easily be reached.
13. Check that the final loaded PCB will fit into its case as designed and all support holes and components are correctly positioned and orientated.