Chapter 6
Printed Circuitry
April 05, 2010

Printed Circuit Board

- A printed circuit board, or PCB, is used to mechanically support and electrically connect electronic components using conductive pathways, or traces, etched from copper sheets and laminated onto a non-conductive substrate.

http://video.google.com/videoplay?docid=-1916931439952544279&q=PCB+board+manufacture&total=19&start=0&num=10&so=0&type=search&plindex=2

Schematic or Logic Diagrams

- Schematic and logic diagrams are used to design the printed circuit board, (pcb) with the list of components.
- The schematic will provide the power and signal paths for the individual components.

Advantages of Printed Circuits

- Can be automatically assembled
- Can be consistently produced without variations in quality
- Lower production costs, (applies for large production runs)
- Reduce the size and weight of finished product
- The circuit’s capacitance will not vary from circuit to circuit.
- Their plug in capability makes assembly and service easy.

Design Data

- In addition to the schematic and the components list, the following additional information must be provided to the pcb designer.
  - Voltage and amperage rating
  - Heat sink requirements for heat dissipating components
  - Grounding requirements
  - Board size and mounting requirements
  - Layer requirements
  - Schematic / logic diagram

Designing the PCB Board

- Primary Considerations
  - Components size.
  - Method by which components will be mounted to the board.
  - Size necessary for mounting holes.
  - Diameter of solder pads
  - (Solder pads are used to provide a surface for the technician to attach the components of the circuits. Size is dictated by the diameter of the component lead wires)
Component Details

Component Dimensions

Directional Components

PCB Design

PC Board Design

- Considerations
  - The width and spacing of conductors must be carefully chosen.
  - Spacing is dictated by the amount of voltage in the circuit, the altitude where it will be used, and the board’s coating.

Width of Conductor

Note: The width of the conductor is determined by the amperage causing heat in the circuit.
Component Location

- Components that will be used at higher altitudes, the resulting lower barometric pressure increases the possibility of current arcing across gaps.
- As a result at higher altitudes the designer must use a wider spacing for components.
- Coating the board with an insulator will also reduce the possibility of arcing across gaps.

Conductor Spacing Standards

Board Size and Style

- The simplest and least expensive board to manufacture is a single sided board.
- The single sided board will have circuitry on one side and the components on the other side.
- Single sided boards are not suitable for the dense circuitry required for digital or certain analog applications.

Single Sided Circuit Layout

Marking artwork will show locations of components and reference designations.
Multi Sided Printed Circuit Boards

- Allow the designer to maintain gap requirements, and fit more components in a given area.
- Boards with up to 20 layers have been used.

Double Sided Printed Circuit Boards

- When creating a double sided board keep the majority of the components on one side of the board, (component side).
- This will reduce the problem/possibility of having components located over circuitry.
- Once the style of board has been chosen the size must be determined.
- It is helpful to use previous projects as guides.

Double Sided Printed Circuit Boards

Note Components and circuitry on same side

Creating / Designing A Printed Circuit Board

- Start with a layout sketch.
- Once the layout has been determined, a formal layout is to be created.
- Layout must be drawn true to scale and all components must be exactly true to scale.
- Three datums will be used to assure that each layer is properly aligned and measurements are accurate.

Component Layout

Copper or Circuit Side
Formal Component Layout

Note 3: Datum Callouts

Representation of Components

Note: Components are unidirectional

Directional Components

Mounting of Components

Arrangement of Components

• Aligned when possible to make simpler for inspection and assembly.
• Adjustable components should be located in a way that keeps making adjustments simple.
Arrangement of Components

Locating Adjustable Components

Locating Test Points

- Test points should be positioned in a location where they can easily be accessed why the equipment is in operation.

Recommended Hole Size for Different Lead Diameters

<table>
<thead>
<tr>
<th>Lead Size (Dia.)</th>
<th>Unplated Hole (Dia.)</th>
<th>Hole Diameter for Plating</th>
</tr>
</thead>
<tbody>
<tr>
<td>.0063 to .0113</td>
<td>.014 ± .003</td>
<td>.020 ± .005</td>
</tr>
<tr>
<td>.0126 to .028</td>
<td>.028 ± .003</td>
<td>.030 ± .005</td>
</tr>
<tr>
<td>.0253 to .0320</td>
<td>.040 ± .004</td>
<td>.040 ± .006</td>
</tr>
<tr>
<td>.0359 to .0492</td>
<td>.052 ± .004</td>
<td>.050 ± .006</td>
</tr>
<tr>
<td>.0508 to .0871</td>
<td>.062 ± .004</td>
<td>.062 ± .007</td>
</tr>
</tbody>
</table>

Proper Mounting Terminal Area

Printed Circuit Board Artwork

- The artwork for each layer is to be created in a different color on a separate layer.
- The different colors will make it easier to follow the circuitry lines on each layer

Selecting the proper mounting terminal diameter- will allow for adequate solder area around each terminal connection.
Manufacturing Process

- Smaller companies may create their manufacturing tools by hand.

http://www.youtube.com/watch?v=Q6Wk1qVleQ0

PC Board Structure

- The majority of printed circuit boards are made by bonding a layer of copper over the entire substrate, sometimes on both sides, (creating a "blank PCB").
- The unwanted copper is the removed after applying a temporary mask (eg. by etching), leaving only the desired copper traces.

Manufacturing Process

- The masking is accomplished by “taping”
- Accuracy is critical to within +/- .005.
- There are five basic rules for taping.
  - Avoid sharp corners, (they cause problems when etching, soldering and foil delamination)
  - Keep interconnections as short as possible.
  - Keep the tapes and media clean.
  - Spread the circuitry out so it is evenly spaced,(this prevents localized heating)
  - Make the terminals larger then the interconnecting lines.

Etching Process

- The etching is accomplished with a photographic process.
- The board is composed of an insulating material and is called substrate.
- Copper foil has been laminated to the substrate.
- A photo resist coating has been applied and will harden when exposed to UV light.
Etching Process

- Prior to exposing the board to UV light the holes must be drilled.
- Once the board has been exposed, (the artwork is a negative control) to the UV light, the exposed photo resist layer will be hardened.
- The unhardened resist will be washed away and the exposed copper will be eaten away by the etching chemicals.

Manufacturing Process

- Connectors that will be pluggable edge connectors must be plated. Typically plated with gold because it has excellent current carrying ability and a high resistance to corrosion.
- Tinning and coating, (Decreases oxidation problems)
- Trim the board to the final size
- Inspection for flaws.

Printed Circuit Drawings

- Logic or schematic drawing
- Layout drawing
- Artwork
- Drill and trim drawing
- Assembly drawing
- Optional drawings (silkscreen/solder mask)

Printed Circuit Board

- Create the assembly drawing on page 122 with the parts list. Use the dimensions provided for the board on page 121 and the component dimensions found on page 127. Locate the components on the board by scaling the drawing on page 122. Accuracy for location should be to within +/- 1/8”