Circuit Board Design

Before layout:
1. Obtain all parts or get physical dimensions from datasheets
2. Create a schematic
   - Put notes on schematic that pertain to the layout
     i. Ground shields
     ii. Component requirements
   - Voltage map for every node
   - Current map for components
3. Determine size of PCB by laying out components onto a sheet of paper
   - Include board mounting method / enclosure
   - Place fixed location parts first – then place remaining parts
   - Allow easy access for user interface
   - Allow room for heat sinks and warm components

Layout:
1. Always work in inches
2. Determine how many layers the board requires
3. Determine layout styles to use
   a. SMT
   b. Thru-hole
   c. Landing Pads
4. Optimize layout by creating short length traces
5. Draw traces on 90 degree or 45 degree angles
6. Follow voltage map for trace to trace spacing
   - Milling bit is 0.012” in diameter – this creates minimum trace to trace spacing
7. Follow current map to make desired traces as large as necessary
8. Allow adequate annular ring around wires and other external connections
9. Design and test any fuses created on PCB traces
10. Remember IC sockets can only have connecting traces on the bottom side of the PCB.
11. Allow extra room to hand solder SMD parts
12. Print out layout and places components on paper – check for problems

Software:
1. Cadence schematics can create a netlist for PCB layout
   a. Netlists link components to aid layout design
2. Components are typically “designed” in part libraries
   a. Don’t assume the library is correct!
3. You can create your own custom components, use the libraries, or just place pads and copper on the PCB to suit the needs of a part.
   a. Revisions are hard to do without created components
Software Output files – to create milled PCB:

Gerber files generated from a PCB layout:

- **Pcb_name.TOP**  Top Copper Layer  (if layer is used)
- **Pcb_name.BOT**  Bottom Copper Layer
- **Pcb_name.BRD**  Board outline layer  (includes any internal board cuts)
- **Pcb_name.DRL**  Drill file

Other things to know:

Drill bit tolerance is typically +/- 0.003”
- Maximum lead diameter < minimum hole size
- Typically allow 0.010” extra hole clearance

Thickness of copper (oz. per sq. ft.):

- ½oz  0.0007”
- 1oz  0.0014”  (prototype material uses 1oz)
- 2oz  0.0028”

Standards / Guidelines for PCBs:
- IPC - American National Standardization Institute (ANSI)
- UL – Underwriters Laboratory
Current Sizing for traces

The melting point of Copper is 1083°C.

The trace width is calculated as follows:

First, the Area is calculated:

\[
\text{Area} = \frac{\text{Current}}{k \cdot (\text{Temp}_\text{Rise})^b} \cdot (1/c)
\]

Then, the Width is calculated:

\[
\text{Width} = \frac{\text{Area}}{\text{Thickness} \cdot 1.378}
\]

For IPC-D-275 internal layers: \( k = 0.0150, \ b = 0.5453, \ c = 0.7349 \)

For IPC-D-275 external layers: \( k = 0.0647, \ b = 0.4281, \ c = 0.6732 \)
## Trace Spacing Guidelines

<table>
<thead>
<tr>
<th>Voltage Between Conductors (VDC or Peak)</th>
<th>Minimum Spacing (inches)</th>
<th>Bare Board</th>
<th>Assembly</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>B1</td>
<td>B2</td>
</tr>
<tr>
<td>0 thru 15</td>
<td></td>
<td>.004</td>
<td>.025</td>
</tr>
<tr>
<td>16 thru 30</td>
<td></td>
<td>.004</td>
<td>.025</td>
</tr>
<tr>
<td>31 thru 50</td>
<td></td>
<td>.004</td>
<td>.025</td>
</tr>
<tr>
<td>51 thru 100</td>
<td></td>
<td>.004</td>
<td>.025</td>
</tr>
<tr>
<td>101 thru 150</td>
<td></td>
<td>.008</td>
<td>.025</td>
</tr>
<tr>
<td>151 thru 170</td>
<td></td>
<td>.008</td>
<td>.050</td>
</tr>
<tr>
<td>171 thru 250</td>
<td></td>
<td>.008</td>
<td>.050</td>
</tr>
<tr>
<td>251 thru 300</td>
<td></td>
<td>.008</td>
<td>.050</td>
</tr>
<tr>
<td>301 thru 500</td>
<td></td>
<td>.010</td>
<td>.100</td>
</tr>
<tr>
<td>More than 500</td>
<td></td>
<td>.0001/Volt</td>
<td>.0002/Volt</td>
</tr>
</tbody>
</table>

**B1** - Internal Conductors  
**B2** - External Conductors, uncoated, sea level to 10,000 ft.  
**B3** - External Conductors, uncoated, over 10,000 ft.  
**B4** - External Conductors, with permanent polymer coating (soldermask).  
**A5** - External Conductors, with conformal coating over assembly.  
**A6** - External Component lead/termination, uncoated.  
**A7** - External Component lead/termination, with conformal coating.