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

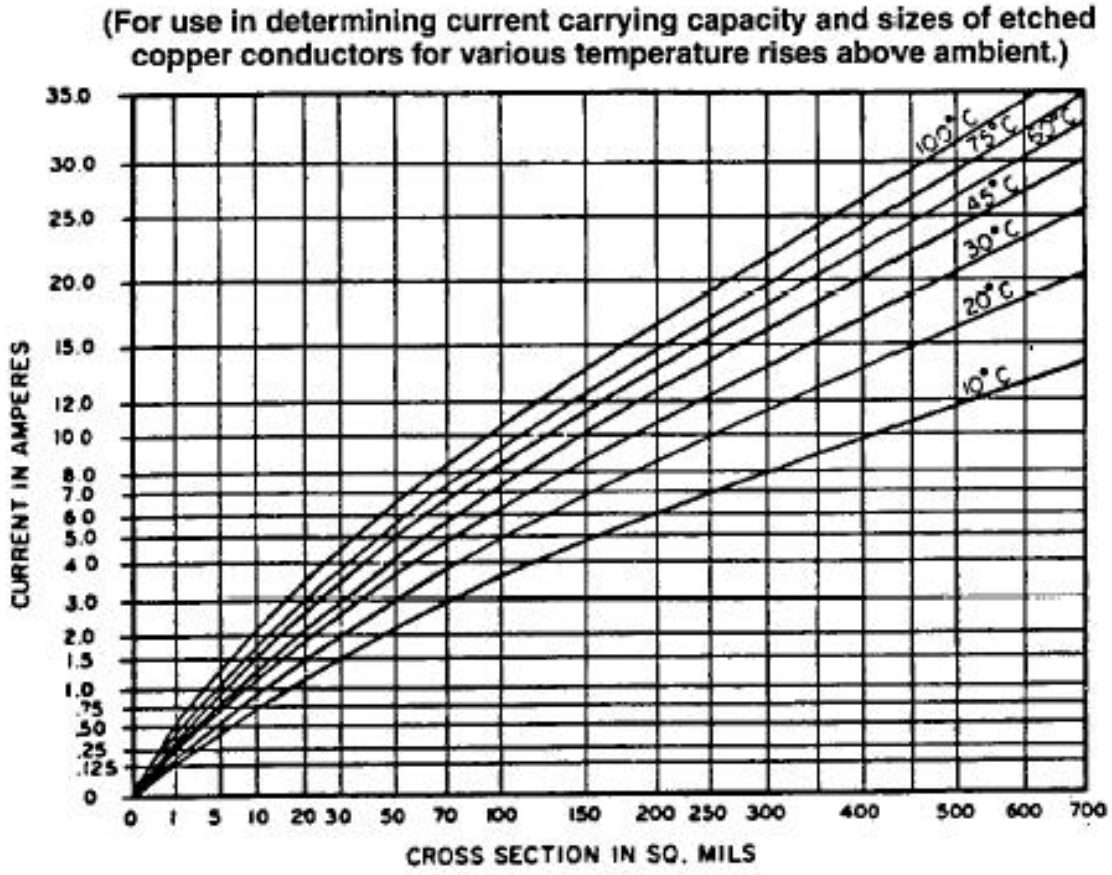

Figure A External Conductors

The **melting point** of **Copper** is 1083C

The trace width is calculated as follows:

First, the Area is calculated:

Area = (Current/(k*(Temp_Rise)^b))^(1/c)

Then, the Width is calculated:

Width = Area/(Thickness*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

Voltage Between Conductors (VDC or Peak)	Minimum Spacing (inches)						
	Bare Board				Assembly		
	B1	B2	B3	B4	A5	A6	A7
0 thru 15	.004	.025	.025	.005	.005	.005	.005
16 thru 30	.004	.025	.025	.005	.005	.010	.005
31 thru 50	.004	.025	.025	.005	.005	.015	.005
51 thru 100	.004	.025	.060	.005	.005	.020	.005
101 thru 150	.008	.025	.125	.015	.015	.030	.015
151 thru 170	.008	.050	.125	.015	.015	.030	.015
171 thru 250	.008	.050	.250	.015	.015	.030	.015
251 thru 300	.008	.050	.500	.015	.015	.030	.015
301 thru 500	.010	.100	.500	.030	.030	.060	.030
More than 500	.0001 /Volt	.0002 /Volt	.001 /Volt	.00012 /Volt	.00012 /Volt	.00012 /Volt	.00012 /Volt

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.