The thickness of the copper on the PCB is nominally specified in ounces per square foot, with 1oz copper being the most common. You can order other thicknesses like 0.5oz, 2oz and 4oz. The thicker copper layers are useful for high current, high reliability designs.

The calculations to figure out a required track width based on the current and the maximum temperature rise are a little complex. They can also be quite inaccurate, as the standard is based on a set of non-linear graphs based on measured data from around half a century ago. These are still reproduced in the IPC standard.

A handy track width calculator program can be found at www.ultracad.com/calc.htm, and gives results based on the IPC graphs.

As a rule of thumb, a 10degC temperature rise in your track is a nice safe limit to design around. A handy reference table has been included in this article to give you a list of track widths vs current for a 10degC rise. The DC resistance in milli ohms per inch is also shown. Of course, the bigger the track the better, so don't just blindly stick to the table.

Track Width Reference Table (for 10deg C temp rise). Track Width is in Thous (mils)			
Current (Amps)	**Width for 1oz**	**Width for 2 oz**	**milli Ohms/Inch**
1	10	5	52
2	30	15	17.2
3	50	25	10.3
4	80	40	6.4
5	110	55	4.7
6	150	75	3.4
7	180	90	2.9
8	220	110	2.3
9	260	130	2.0
10	300	150	1.7

Pads

Pad sizes, shapes and dimensions will depend not only upon the component you are using, but also the manufacturing process used to assemble the board, among other things. There are a whole slew of standards and theories behind pad sizes and layouts, and this will be explained later. Suffice it to say at this stage that your PCB package should come with a set of basic component libraries that will get you started. For all but the simplest boards though, you'll have to modify these basic components to suit your purpose. Over time you will build up your own library of components suitable for various requirements.

There is an important parameter known as the pad/hole ratio. This is the ratio of the pad size to the hole size. Each manufacturer will have their own minimum specification for this. As a simple rule of thumb, the pad should be at least 1.8 times the diameter of the hole, or at least 0.5mm larger. This is to allow for alignment tolerances on the drill and the artwork on top and bottom layers. This ratio gets more important the smaller the pad and hole become, and is particularly relevant to vias.

There are some common practices used when it comes to generic component pads. Pads for leaded components like resistors, capacitors and diodes should be round, with around 70 thou diameter being common. Dual In Line (DIL) components like IC's are better suited with oval shaped pads (60 thou high by 90-100 thou wide is common). Pin 1 of the chip sould always be a different pad shape, usually rectangular, and with the same dimensions as the other pins.

Most surface mount components use rectangular pads, although surface mount SO package ICs should use oval pads. Again, with pin 1 being rectangular.

Other components that rely on pin numbering, like connectors and SIP resistor packs, should also follow the "rectangular pin 1" rule.