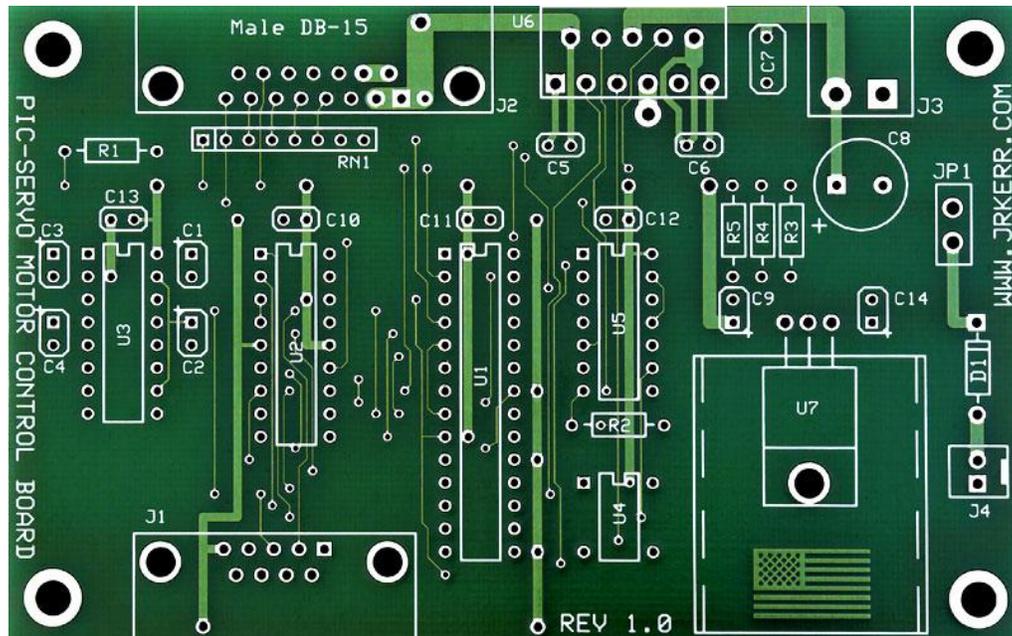


# Application Note:

## PCB Design

By: Wei-Lung Ho



### Introduction:

A printed circuit board (PCB) electrically connects circuit components by routing conductive traces to conductive pads designed for specific components supported by a non-conductive substrate. PCBs usually contain more than one layer, but can contain as many layers as desired at a higher cost. Multilayer boards allow for more component placement flexibility by using vias which are plated through-holes that allow interconnection of components across different layers of the design board, since advanced PCB architectures may contain circuit components embedded within the layers of the PCBs.

Although PCBs require additional design effort to lay out the circuit components, but this additional PCB work allows automated manufacturing and assembling. Automated manufacturing and assembly enables complicated circuit architectures to be built accurately. This minimizes human errors such as misrouting wires when circuits are constructed by hand. Overall, designing and manufacturing PCBs are faster and cheaper than other methods of circuit fabrication.

PCBs are designed using Electronic Design Automation (EDA) tools which are software made to assist in fabricating PCBs. EDA tools interactively assist the designer in following conditions such as design rules that can be set by the designer or component overlapping that can occur due to attempts in minimizing the costs of fabrication. However a major downside to using EDA tools is that these tools can cost thousands of dollars to purchase. A commonly used industry standard PCB design software by Cadence Allegro cost about \$5000 USD, however there are also free PCB design tools such as KiCAD and Eagle that can be used to design PCBs, but come at a cost. Lower cost and free PCB design tools come with fewer features compared to Cadence Allegro.



The footprint assignment is like the interior designing portion and this step assigns the actual parts of the component to its physical dimensions. Most PCB designing software is usually accompanied by a default library which contains commonly used footprints of circuit components. Some common footprint for components such as resistor, capacitor and inductors are surface mounts (SMD) of sizes such as 0402, 0603 and 0804 where 0X and 0Y referring to the physical sizes of the components.

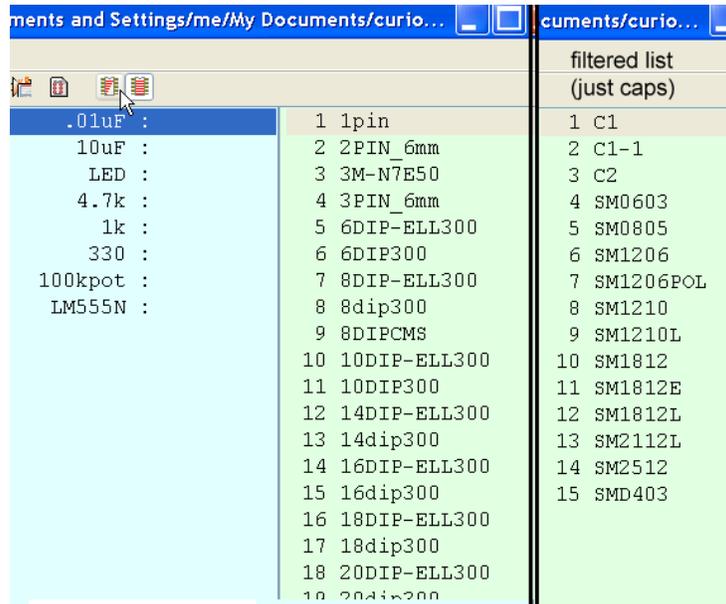


Figure 2: Footprint assignment.

However, on many occasions the desired component will not be found in the default library which means that they will have to be drawn using some kind of component editor or maker that comes with the PCB design software. In general, it is good practice to draw all the needed components even if it can be found in the library to ensure that the physical dimensions and pin layout match the exact dimensions given on the manufacturer's datasheet.

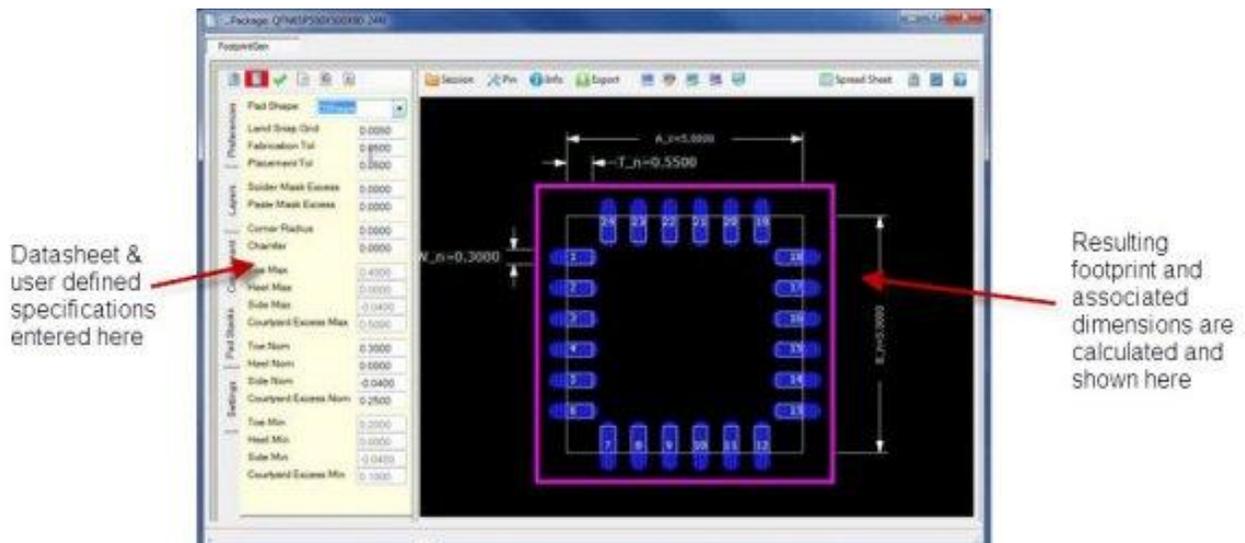


Figure 3: PCB component editor.

The final step in designing PCB's is the PCB Layout. Here, the assembled footprints are arranged in such a way that the physical board minimizes its total size. The reason minimizing the physical dimensions of the printed circuit board is important because people want to reduce the cost of purchasing each board. The term used to for the copper connections on PCB's is called a trace. In this step, there are many tricks to minimizing the size of the PCB layout.

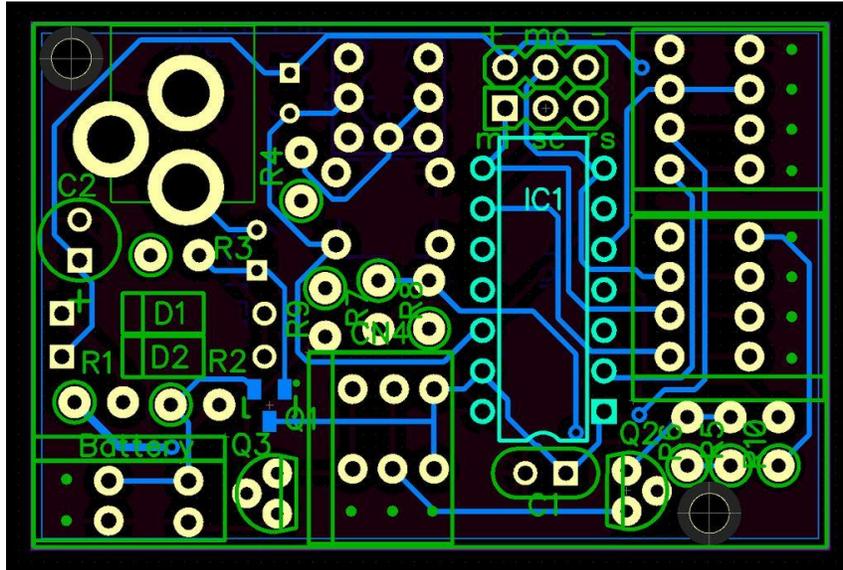


Figure 4: PCB Layout.

A common technique for drawing traces if the traces have to cross each other, is to drop a via which takes the trace to the bottom side of the board (assuming this is a 2-layered board), and then route it to a convenient location to connect the components together. In the image below, the hole that begins at the top side of the board and terminates on the bottom side is a via hole. In consideration of board layers, a PCB can have as many layers as desired. Having more layers in a PCB greatly reduce the difficulty of connecting the components, however the difficulty of producing a multilayer board increases the price of the board, so it is best to use as few layers as possible when designing the layout of PCB.

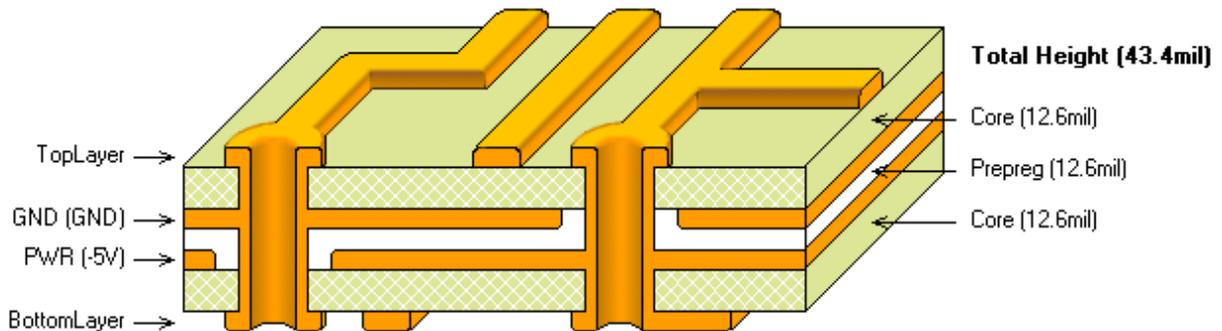


Figure 5: Trace and via holes.

### High Frequency Considerations:

When designing a PCB that transmits high frequency signals, impedance controlled transmission lines must be taken into consideration. RF components are generally matched to  $50\Omega$  or  $75\Omega$ , meaning that each trace that connects one RF component to another is either  $50\Omega$  or  $75\Omega$ , where the impedance of the matched transmission line is known as the characteristic impedance. The reason RF systems must be matched is because impedance mismatches transmitting high frequency signals cause signals to be reflected back toward the signal source which results in a loss of power.

An RF system is usually matched to a characteristic impedance using some type of microstrip line structure. A microstrip line is a conductive routing path used as a trace that contains a fixed width. All microstrip lines have a different width for a different frequency, dielectric thickness, trace thickness, dielectric material, and more. In designing just the impedance matching portion of the RF system, there are many considerations. In addition to choosing a microstrip line width, the designer must also consider the type of microstrip structure that would best accommodate the PCB that would minimize the size of the overall board. As a general rule of thumb, RF traces using microstrip lines should be kept as short as possible to minimize the parasitic effects that follow the nature of the electromagnetic phenomenon such as mutual inductance and capacitive coupling.

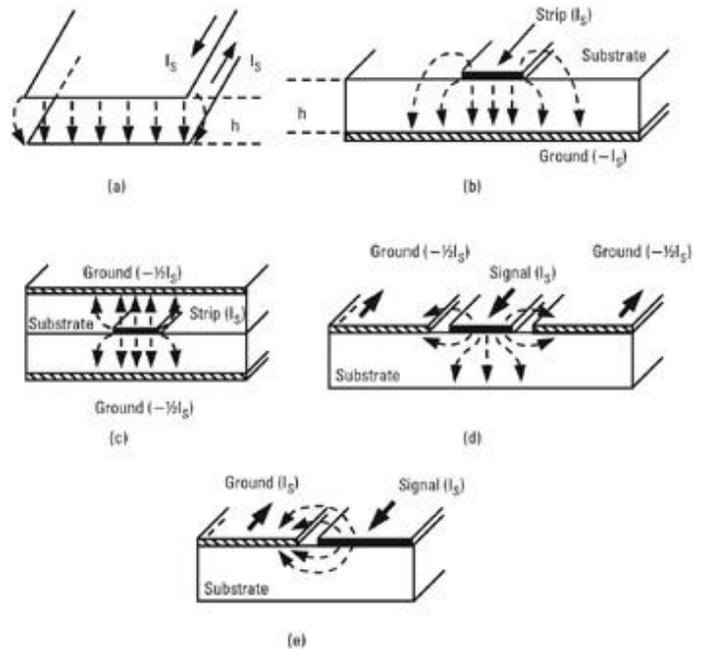


Figure 6: Types of microstrip lines with the electromagnetic effects. (<http://www.globalspec.com/reference/61041/203279/3-5-planar-transmission-lines>)

In addition to matching the system to a characteristic for carrying the signal trace, electromagnetic phenomena are also controlled by the substrate's dielectric constant which is determined by the substrate material as well as its dimensions. Things like the substrate material and substrate thickness play an important role in dictating the dimensions of a certain microstrip structure implemented to carry the high frequency signal. Substrate material containing a higher dielectric constant generally reduces the thickness of the PCB structure and width of the microstrip line. Although higher dielectric substrates reduce the cost based on the size of the board, higher dielectric material will cost more per area than a lower one. As the operating center frequency gets higher, material with a higher dielectric constant is desired to minimize the loss accumulated within the dielectric material and reduce the overall size of the PCB. However, dielectric material used to fabricate PCB's cannot be arbitrarily chosen. In choosing a PCB substrate, the desired material should be chosen based on the manufacturer's specifications. In using the manufacturer's specifications, an accurate calculation of the microstrip structure's dimensions can be maximized.

| Media            | Dielectric    | Layer Thickness in mils (mm)      | Center Conductor in mils (mm) | Gap       | Characteristic Impedance |
|------------------|---------------|-----------------------------------|-------------------------------|-----------|--------------------------|
| Microstrip       | Prepreg (3.8) | 6 (0.152)                         | 11.5 (0.292)                  | N/A       | 50.3                     |
|                  |               | 10 (0.254)                        | 20 (0.508)                    |           | 50.0                     |
| Diff. Pair       | Prepreg (3.8) | 6 (0.152)                         | 25 (0.635)                    | 6 (0.152) | 50.6                     |
| Stripline        | FR4 (4.5)     | 12 (0.305)                        | 3.7 (0.094)                   | N/A       | 50.0                     |
| Offset Stripline | Prepreg (3.9) | 6 (0.152) upper, 10 (0.254) lower | 4.8 (0.122)                   | N/A       | 50.1                     |
| Coplanar WG      | Prepreg (3.8) | 6 (0.152)                         | 14 (0.35)                     | 20 (0.50) | 49.7                     |

Figure 7: Dimensions of some microstrip structures of different material.

In designing the PCB layout, the recommended practice is to use a solid, continuous ground plane on the bottom side of the PCB assuming the topside is used to layout the circuit components. This ground plane on the bottom side of the PCB must not be broken under long transmission line routing because this introduces parasitic capacitances in the PCB. In the figure to the right, we can see that there are many transmission lines on the ground plane, however these lines are short enough to where they will not introduce parasitic effects into the RF system. The continuous ground plane helps prevent ground inductance due to the ground current return paths. In addition to the ground plane, ground vias should be placed liberally to prevent cross-coupling from RF and other signal lines throughout the PCB.

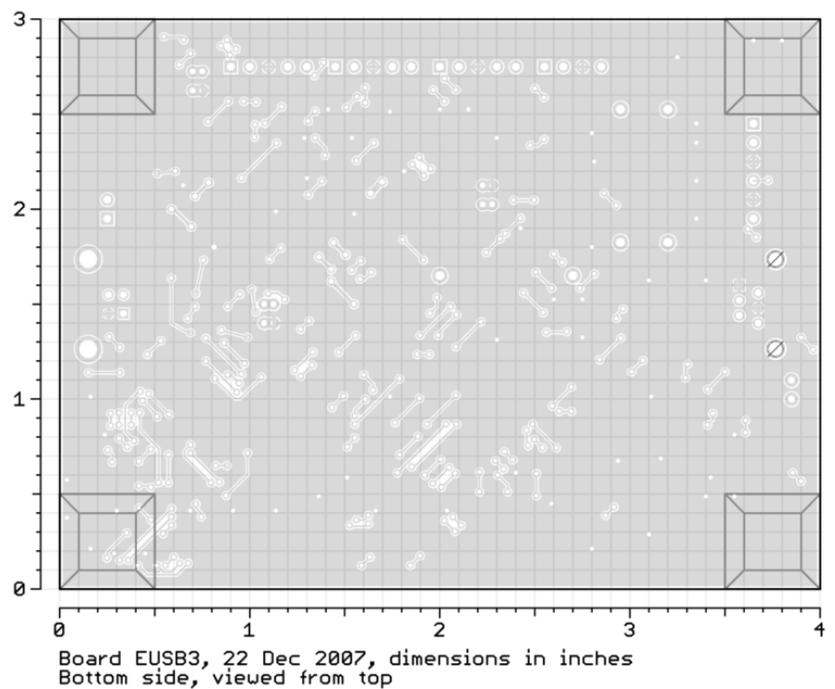
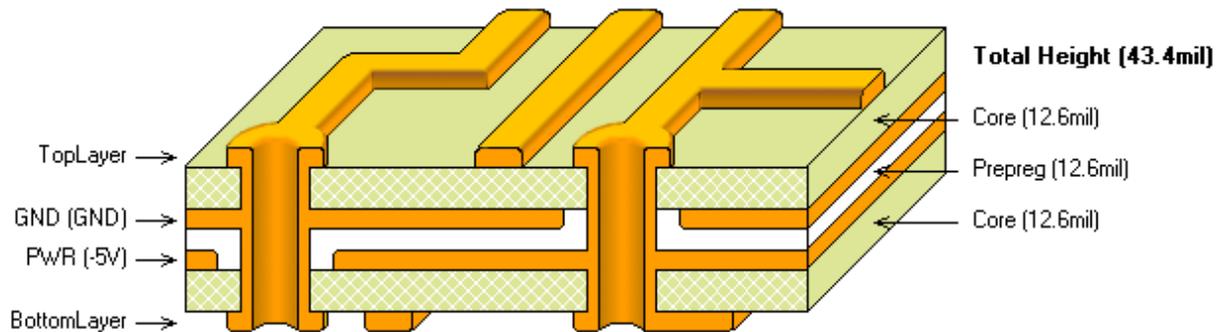


Figure 8: PCB layout with a continuous ground plane.

In high frequency PCB design, signal isolation is an important concept to prevent unintended coupling between transmission lines. One thing to keep in mind is to keep RF traces as far away as possible and should not be routed nearby each other for extended distances. Transmission line coupling decreases as the separation between the lines increase. For multilayer boards, if lines must cross each other on separate layers, a ground plane should be inserted between the signal carrying layers. Signal lines carrying high power levels should be kept away from other lines as much as possible.  $V_{CC}$  and power lines should be routed on dedicated layers to reduce the signal noise of the system.



For components that are connected in shunt or grounded components like decoupling capacitors, a recommended practice is to use at least two ground vias for each component. Using multiple ground vias helps reduce the effect of via parasitic inductances. Via grounds islands can be used for groups of shunt-connected components.

In addition to via islands, some IC's may require solid ground plane on the component layer directly underneath the IC. The ground plane serves two purposes for the IC component, the first purpose is to carry the DC and RF currents through the PCB to the ground. The second purpose is to provide a thermal heat sink to the component, so the maximum number of vias that can be accommodated should be taken into consideration by the layout. If available, the vias should be filled with thermally conductive paste to enhance the capability of the heat sink.

